Image Cover Sheet

CLASSIFICATION	SYSTEM NUMBER 144581
UNCLASSIFIED	

TITLE

SIMPLIFIED MODELS FOR SHIP ANALYSIS

System Number:

Patron Number:

Requester:

Notes: REQUEST

DSIS Use only:

Deliver to: RL

UNLIMITED DISTRIBUTION

National Defence
Research and Development Branch

Défense nationale
Bureau de recherche et développement

DREA CR/93/428

SIMPLIFIED MODELS FOR SHIP ANALYSIS
FINAL REPORT

by
M.E. Norwood et al

MARTEC Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia, Canada
B3J 3J8

CONTRACTOR REPORT

Prepared for

Defence Research Establishment Atlantic

Centre de Recherches pour la Défense Atlantique

DEFENSIO PER SCIENTIAM

Canada

PLEASE DIRECT ENQUIRIES TO:

THE CHIEF,
DEFENCE RESEARCH ESTABLISHMENT ATLANTIC,
PO BOX 1012,
DARTMOUTH, NOVA SCOTIA,
CANADA.

UNLIMITED DISTRIBUTION

National Defence
Research and Development Branch

Défense nationale
Bureau de recherche et développement

DREA CR/93/428

SIMPLIFIED MODELS FOR SHIP ANALYSIS
FINAL REPORT

by
M.E. Norwood et al

MARTEC Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia, Canada
B3J 3J8

Scientific Authority ______ Neil Pegg

W7707-6-7739
Contract Number

December 1992

CONTRACTOR REPORT

Prepared for

Defence Research Establishment Atlantic

DEFENSIO PER SCIENTIAM

Centre de Recherches pour la Défense Atlantique

Canada

ABSTRACT

This report describes the work carried out under contract to the Defence Research Establishment Atlantic (DREA), Contract Serial No. OAE.86-0042, DSS File No. W7707-6-7739/01-OAE. This report is supported by two other documents produced under this contract (References 1 and 2).

The purpose of this work was to produce, through research and development, a capability for simplified analysis of ship structures. Simplified techniques for generating input data for finite element analysis have been developed. The work performed is described in Sections 2 to 12.

RÈSUMÈ

Le présent rapport décrit des travaux exécutés à contrat pour le Centre de recherche pour la défense (Atlantique), numéro de série du contrat OAE.86-0042, dossier no W7707-6-7739/01-OAE du MAS. Ce rapport s'appuie sur deux autres documents rédigés en vertu du même contrat (references 1 et 2).

Le but de ces travaux consistait à produire par la recherche et le développement un instrument d'analyse simplifiée de structures de navires. Des méthodes simplifiées de production de données d'entrée pour l'analyse par la méthode des éléments finis ont été mises au point. Les travaux exécutés sont décrits aux sections 2 à 12.

TABLE OF CONTENTS

TABLE OF CONTENTS (Continued)

1.0 INTRODUCTION

This report describes the work carried out under contract to the Defence Research Establishment Atlantic (DREA), Contract Serial No. OAE.86-0042, DSS File No. W7707-6-7739/01-OAE. This report is supported by two other documents produced under this contract (References 1 and 2).

The purpose of this work was to produce, through research and development, a capability for simplified analysis of ship structures. Simplified techniques for generating input data for finite element analysis have been developed. The work performed is described in Sections 2 to 12.

2.0 Simplifying Modelling Options

2.1 Hybrid Beam Element

The capability to model a beam-plate combination as a hybrid beam where a specified effective breadth of plate is assumed to act with the beam was provided in VAST.

A description of the element can be found in Reference 3. The element is termed 'hybrid' because the axial stiffness and the bending stiffness are based on different cross-sections. The axial stiffness is based on the area of the beam alone, while the bending stiffness is based on the combined beam section and effective width of plate.

The hybrid beam element is offered as a special form (IFORM=1) of the general beam element in VAST (IEC=3). The proper usage of the element is shown in Figure 2.1. The section properties of the actual beam section is defined as usual. The user specifies an effective width of the plate (BEFF), the plate thickness (TP), the distance from the plate to the centroid of the beam section (DP), and an orientation code (IORC).

When using this element, it is assumed that the plate elements are located such that their membrane stiffness acts in the plane of the neutral axis of the combined beam-plate section. In this way, the plate membrane stiffness and beam axial stiffness combine to make up the total in-plane stiffness of the structure. The effective plane of action is therefore the plane passing through the combined centroid.

Beam stresses are computed in the standard way. The bending component of the beam stress is computed using the moment of inertia of the combined beam section and effective width of plate. The stress component in the plate elements due to beam bending is not included in

the plate stress output in VAST. This component is approximately equal to the beam fiber stress at the beam-plate interface and should be added to the appropriate plate stress component provided in the VAST output.

Hughes (Reference 3) shows that for hull module analysis, the hybrid beam element can yield more accurate stresses than the more commonly used eccentric beam element. Furthermore, since the effective breadth of plate approach allows for shear lag, the necessity to use high-order plate elements or more than one low-order plate element between beams is avoided. Of course, the accuracy of the hybrid beam depends on the choice of the effective breadth, which usually lies in the range between 50% and 80% of the actual beam spacing.

2.2 Stiffened Membrane and Plate Elements

The ability to model stiffened panels by smearing longitudinal and/or transverse stiffeners into an equivalent membrane or plate thickness was provided in VAST. This feature is offered in two elements in VAST, the 4-noded shell element (IEC=5) and the 4-noded membrane element (IEC=11). The latter is a modified version of the old 4-noded warped/stiffened membrane element.

The stiffened panel option is offered as a special form (IFORM=1) of element types IEC=5 and 11. If the user sets IFORM equal to zero (default) for either element type, the formulation will proceed on the basis that no stiffeners exist in the particular element group. However, if the user sets IFORM=1, it will be assumed that some or all elements in the particular group have stiffeners. A typical arrangement of stiffeners is shown in Figure 2.2.

For each element group containing stiffeners, the user must provide additional data, including a table of stiffener properties, and for each element within the group, an indication of whether longitudinal and/or

transverse stiffeners exist, and orientation angles with respect to the local x-axis (ϕ_1 and ϕ_2 in Figure 2.2).

The table of stiffener properties provides, for each stiffener type, its pitch (see Figure 2.2), its cross-sectional area, and the Young's modulus and mass density for the stiffener material. For element type IEC=5 (which has bending), an area moment of inertia can also be supplied. If supplied (i.e. a non-zero value is entered), only the stiffener effect on bending will be modelled. (At present, stiffener inplane and bending effects cannot be combined.)

Prior to the current work, element type IEC=11 in VAST was termed "Stiffened Warped Orthotropic Membrane". Its formulation with respect to stiffener representation involves modelling the stiffener/plate combination as a membrane with orthotropic material. Under the current contract, this same concept for modelling stiffeners was incorporated in element type IEC=5.

Since its incorporation into VAST, the "Stiffened Warped Orthotropic Membrane" has received little usage. And, although it can serve as a general 4-nodel quadrilateral, it has never been used for this purpose. This is probably due to the fact that the element was rather awkward to use, especially in unstiffened situations. Under the current contract, this situation was rectified. First, the elements name was changed to "4-Noded Quadrilateral Membrane". Second, the default formulation is now for an unstiffened general 4-noded quadrilateral membrane element (i.e. IFORM=0). Third, the input data is greatly simplified for unstiffened situations.

2.3 Pressure Loads on Hull Due to Balancing on a Wave

The ability to generate and apply pressure loading on a full hull model due to balancing on a wave was developed. This was done in

conjunction with Mr. S. Ando at DREA. The procedure is based on a merging of DREA program POSBOW (Reference 4) and the HVAST program HLOAD2, the latter providing the ability to calculate pressure loads on a ship's hull using transverse pressure curves at specified sections.

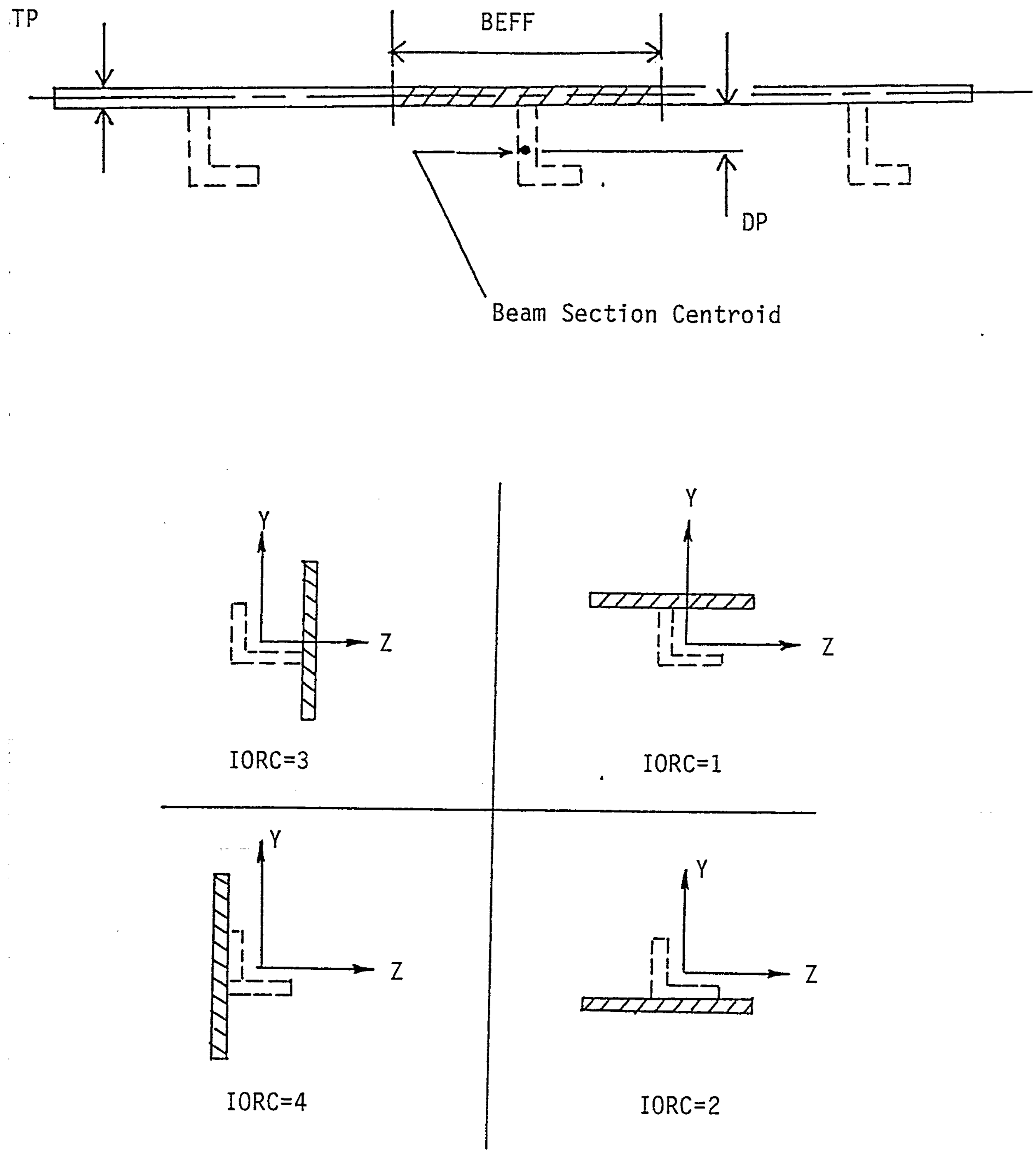


FIGURE 2.1: Hybrid Beam Element

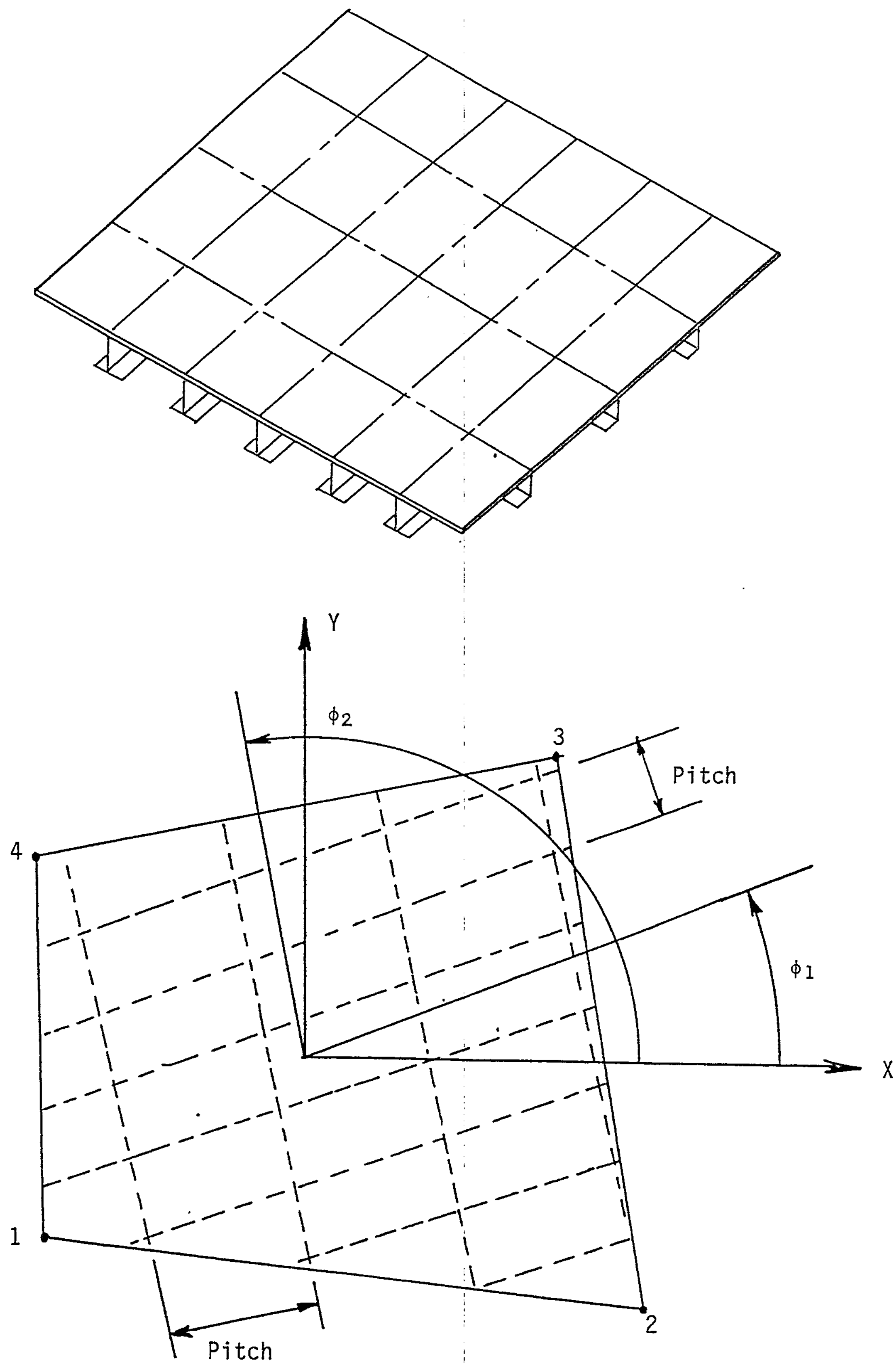


FIGURE 2.2: Typical Arrangement of Stiffeners in Element Types IEC=5 and 11

3.0 IDENTIFICATION OF BOUNDARY CONDITIONS

The ability to identify nodes, interactively through the use of a graphics cursor on a display of the model was developed. A special program called VASTBC was developed for this purpose. A description of the program, its capabilities and operating instructions, as developed under this contract are given in Appendix A. A more recent version of VASTBC produced under other contracts is described in Reference 5.

4.0 GENERATION OF NODAL POINT EDGE LOADS

The ability to generate nodal point edge loads on finite element models of stiffened plates and shells was developed. A special program called HULLD1 was developed for this purpose. A description of the program, its capabilities and operating instructions are given in Appendix B.

5.0 BOUNDARY ELEMENTS FOR PLATE BENDING PROBLEMS

The boundary element method (BEM) is the discrete form of the boundary integral equation method (BIEM) which can be applied to solve boundary value problems in engineering and science. BEM has been used to solve problems in finite regions of arbitrary geometry as well as in infinite regions. The method was initially applied to linear problems but several types of nonlinear problems have been successfully solved in recent years. BEM has truly evolved into a powerful general-purpose analysis procedure. It possesses some distinct advantages over domain techniques such as the Finite Difference Method (FDM) and the Finite Element Method (FEM) for a wide class of boundary value problems.

The analysis of plates and shells under static and dynamic loads is of great interest to mathematicians, engineers and naval architects both from the theoretical and practical viewpoint. This section of the report is devoted to the potential application of the BEM to the analysis of plates and shell structures.

5.1 Historical Comments on BEM Development

The boundary integral equation method developed from the work of several Russian mathematicians including Muskhelishvili [7], Miklin [8], Kupradze [9] and Smirnov [10] but was not very popular with engineers because of the unfamiliar mathematics associated with it. This method was mainly used in fluid mechanics and general potential problems where it was referred to as the "source" method. As such, it was an "indirect" method in the sense that the unknowns were not physical variables of the problem. Work on this method continued throughout the 1960's and 1970's with Jaswon [11], Symm [12], Massonet [13], Hess [14] and others. The "direct" form of the boundary integral equation method originated with the work of Cruse and Rizzo [15] in elastostatics. The distinguishing feature of the direct method is it is based on the physical variables of

interest to the analyzer. So-called "fundamental" solutions which (in elastostatics) represent Kelvin's solution for a point load in an infinite body must be available. The load on a body is generally more complex and for this reason, displacements and stresses at any point within the problem domain are then determined by superimposing fundamental solutions for a number of such point loads.

Applications of the BEM has gained momentum in recent years. In solid mechanics, the method has been applied to most types of linear problems. This research includes linear elasticity [16, 17], linear viscoelasticity [18], thermoelasticity [19], linear elastic fracture mechanics [20-24], elastic torsion [25], bending of elastic plates [26], shell theory [29], and wave propagation [28-30]. Applications to non-linear problems are of more recent vintage. The first formulation for time-independent plasticity was that due to Swedlow and Cruse [31]. It accounted for strain hardening effects. No numerical results were given and it was Mendelson [32] and Mendelson and Albers [33] who validated the methodology by producing numerical results for some elastoplastic problems. Improved numerical implementations were later supplied by Riccardella [34] and Cruse [35] who assumed displacements and tractions to vary linearly over elements. Another formulation for plasticity using an equivalent body force based on initial stress is due to Banerjee and Mustoe [36]. Mukherjee and coworkers have also applied BEM to nonlinear problems of time-dependent inelastic deformation. Applications of interest to them included viscoplasticity [37,38], inelastic torsion of prismatic shafts [39], inelastic bending of plates [40], and inelastic fracture mechanics [41].

There have been rapid developments in improved numerical techniques applied to boundary element methods, akin to the introduction of isoparametric elements in the finite element method. The work of Boissenot, Lachat and Watson [42-45] illustrate this trend. Parametric representations of geometry and functions were used in their formulation. The

boundary elements were allowed to be curved, with quadratic variation of geometry with respect to the intrinsic coordinates, while displacements and tractions were considered to vary either linearly, quadratically, or cubically with respect to the same coordinates. Other authors such as Cruse and Wilson [46] and Tan and Fenner [47,48] showed that isoparametric elements with geometry and variables defined with the higher order variations in functions (quadratic) permitted high resolution of stresses in three-dimensional problems.

These early publications are important in the sense that they demonstrated that the BEM was a very powerful and accurate numerical technique and they provided a firm foundation for its further development.

5.2 Comparison with Other Numerical Methods in Continuum Mechanics

The underlying aim of any numerical method of solution for an engineering problem is to determine an approximate solution to the governing differential equations when these cannot be solved analytically. The performance of any particular method should be judged by the accuracy of the solutions provided and the amount of effort (cost) involved. Domain methods of the finite difference and the finite element methods are the main rivals of the BEM.

For linear problems at least, BEM has the advantage over the FEM in terms of effort involved in creating mesh data (even when the process is automated) due to the effective reduction in problem dimensionality by one. The problem unknowns are associated with boundary (or surface) discretization in BEM rather with full body discretization in FEM. The same reduction of dimensionality does not extend to nonlinear problems. The interior must be modelled for nonlinear problems albeit in certain cases such as elastoplasticity, the interior modelling can be confined to selected regions (eg. the region surrounding a crack tip). This observa-

tion also supports the suggestion that less unwanted information is produced. Commonly, maximum stresses occur on the boundary so that the determination of internal stresses is unnecessary. Another related advantage is that problems in infinite domains can be solved as easily as those in bounded domains. It should be noted that the matrices resulting from the application of BEM are fully populated but tend to be well conditioned. This arises from the fact that the singular kernels in the integral equations tend to weigh the unknown quantities near a singular point more heavily than those far from it, thus causing the dominant components of the coefficient matrices to be on or near the diagonal. It has also been argued that BEM is more economical with respect to both computer time and storage than its competitors for solutions of the same accuracy. BEM offers continuous interior modelling within the solution domain thus permitting high resolution of stresses and displacements. This is particularly beneficial for stress concentration and fracture mechanics problems. Furthermore, incompressible or nearly incompressible material behaviour, when the value of Poisson's ratio is equal to or nearly equal to one half, can be easily accommodated.

An important limitation of the direct BEM formulation is that the fundamental solutions must be available in an appropriate infinite body having the same material properties as the finite body being analyzed. While fundamental solutions are easily derived for certain differential operators in homogeneous materials, they might be difficult to obtain for more general non-homogeneous materials. While the BEM is good for physically compact problems having small surface-to-volume ratios, the accuracy of its numerical integration deteriorates as the surface-to-volume ratio increases. Furthermore, the BEM is unsuitable if information is required at a large number of internal points.

As demonstrated above, neither the FEM or the BEM are effective for the full range of engineering problems. This suggests the application of a combined finite-boundary element approach in order to exploit the

complimentary advantages of the two methods. Such an approach is expected to be especially effective in infinite or semi-infinite problems representative of geotechnical applications and ocean engineering. Approximate domain truncations are often questionable in these problems due to the slow decay of the solutions and the absence of a natural length parameter by which to gauge the distance at which the relevant physical fields become truly negligible. The coupled approach would involve far field effects being efficiently modelled by the BEM component and the near field details being captured by the FEM component. The coupling of FEM and BEM for static problems was first proposed by Zienkiewicz et al. [49], Shaw et al. [50,51] and Osias et al. [52]. Investigations were continued by other investigators including Kelly et al. [53], Brebbia and Georgiou [54], Margulies [55], Bolteus and Tullberg [56], and Beer and Meek [57,58]. Two major problems are associated with the coupling process. Firstly, the stiffness matrix deduced from the "direct" boundary element method is non-symmetric. Secondly, this stiffness matrix does not fulfill the equilibrium requirements of stiffness matrices, namely that columns should sum to zero. Hartman [59] gave a theoretical discussion on these two problems with stiffness matrix as generated from the direct boundary element method. Tullberg and Bolteus [60] conducted a numerical study on the accuracy of seven different boundary element stiffness matrix formulations which might be contemplated for a combined BEM-FEM approach. Methods of greatest interest involved the direct coupling of the BEM and FEM coefficient matrices. This could be accomplished by various means including: presenting the BEM formulation as a special case of the generalized FEM formulation; forcing the BEM coefficient matrices to be symmetric by using an energy minimization scheme; and presenting the BEM coefficient matrices as stiffness matrices.

5.3 Literature Review on BEM Application to Plate Bending Problems

Stiffened plate and shells are known to exhibit rapid stress variations near boundaries or near connections to stiffeners. Because of demonstrated ability of boundary elements to accurately resolve stress fields with large gradients, the potential application of the BEM to this class of problems was contemplated under this contract. A literature search was conducted and a bibliography containing relevant references has been prepared and is supplied as Appendix H of this report. Although numerous references were located, the list is by no means exhaustive. A short review of the literature collected follows.

There are several different approaches to formulate boundary integral equations for bending of thin elastic plates. Within each approach, variations in the method can lead to significant differences in properties for the resulting integral equations.

Among the earliest formulations proposed for the treatment of plate bending were the indirect methods of Jaswon and Maiti [61] and Maiti and Chakrabarty [62]. A characteristic of these formulations is that natural variables of the problem such as boundary deflection, normal slope, moment and shear are not used. Rather source distribution densities which define harmonic potentials are used and the harmonic potentials are then related to the displacement or its derivatives. An alternate approach was suggested by Altiero and Sikarski [63]. It involves embedding the original problem in a larger plate for which the Green's function is known. On the otherhand, Wu and Altiero [64] and Tottenham [65] proposed placement of the source distribution on a contour outside the boundary and extended the method to include anisotropic material behaviour. Another indirect formulation is due to Hansen [66]. Such indirect methods work well on the specific problems for which these methods were designed but frequently may exhibit poor numerical characteristics when applied to other problems.

In parallel, direct formulations based on a reciprocal work identity were being developed by Forbes and Robinson [67], Bezine [68], Bezine and Gamby [69], Tottenham [65], Stern [70,71], and others. These formulations involve constructing boundary integral equations in terms of natural boundary variables of displacement, normal slope, bending moment and shear. Special singular auxiliary functions called "fundamental solutions" are generally used. The resulting integral equations are generally singular and must be interpreted in a principal value sense. There are some undesirable consequences of the presence of singularities when the equations are discretized for numerical solution. An alternate direct formulation by Stern and Lin [72] avoids the non-singular boundary integral equations with consequent simplications in the numerical implementations.

No publications were located on the application of the boundary element method to stiffened plates or shells. Two approaches appear possible. Firstly, a combined BEM-FEM analysis could be performed with a boundary element implementation for plate bending adopted within a finite element program such as VAST. The finite element program would of course provide a finite element modelling of the beam stiffeners. Alternately, a boundary element implementation of the beam stiffeners could be selected for implementation within a BEM program together with the plate bending capability. A potential beam stiffener formulation has been documented by Butterfield [73].

5.4 Selection of BEM for Plate Bending

The preferred boundary element formulation for bending of thin plates representative of naval structures is that described by Stern and Lin [72]. It is a new direct formulation which utilizes a new multi-valued fundamental solution. The advantage of the new fundamental solution is that the severity of the singularity in the integral equations is reduced and convergence is normal rather than only in the

principal value sense. This simplifies the subsequent numerical treatment.

The boundary curve is required to be everywhere smooth (continuously turning) except for a finite number of corners. The primary variables of the problem are the deflection w, the normal slope N, the boundary moment M, the equivalent shear V at each point of the boundary and the corner forces $F^{(k)}$ at each boundary corner.

In published example problems, this boundary element implementation was demonstrated to exhibit rapid convergence and good coarse mesh accuracy for plate bending problems involving various geometries and load types. Also abrupt changes in element size did not result in any significant deterioration of accuracy. In many other boundary element implementations involving singular equations, abrupt changes in mesh size would have to be avoided unless very elaborate procedures are instituted.

6.0 ASSEMBLY AND TESTING OF HVAST SUITE OF MODELLING AND POST-PROCESSING PROGRAMS

6.1 Organization into Menu Driven System

The HVAST suite of programs were organized into a menu driven system. This system is based on the nine Model Types shown in Figure 6.1 and the eighteen Analysis Steps shown in Figure 6.2. The system has been designed to accommodate a large number of special purpose structural analysis programs for the analysis of ships, developed for DREA, DRES and DMEM, by Martec Limited, in the past 15 years. The DRES programs included relate to airblast and underwater explosion, while the DMEM programs are related to lattice masts of warships.

To help in the organizational task, a consistent naming scheme for existing and future programs was adopted. This name change is documented in Table 6.1. The convection used is as follows:

Symbols	Definition
HV ----	data generator
HG ----	graphics program
-- EB --	equivalent beam
-- 2D --	2-dimensional
-- 3D --	3-dimensional
-- SP --	special purpose
-- LD --	loads
-- LM --	lumped masses
-- AM --	added mass
-- DIS --	displacements
-- STR --	stresses
-- SKC --	skew coordinates
-- LWD --	longitudinal weight distribution

The menu driven control program utilizes the application matrix shown in Figure 6.3. Each program can be transparently activated by selecting desired options from a menu displayed on the terminal screen. A simple modification to the application matrix allows a new module to be

added to HVAST with ease.

The HVAST driver uses a text file called HVAST.TXT. This file contains all information with regard to model types available, analysis steps which can be performed for the various model types, and a dialogue describing in detail each module within HVAST. This file can be edited to incorporate changes at any time, with no changes to the driver. The most recent HVAST.TXT file is provided in Appendix C.

The HVAST driver can be executed in either a computational mode (with minimum, limited or full dialogue), or a review mode (with full dialogue). The latter mode is particularly useful since it gives the user a means of reviewing the capabilities of HVAST or a particular module and the input data required. When relevant, the appropriate location in the HVAST Reference Manual (not available at present) is identified.

All input to HVAST is provided on a PREFX.HVCR file, where keywords are used to identify the section of data pertaining to a particular module. Line printer output is provided on a PREFX.HVLP file.

A terminal session for HVAST for performing an underwater explosion analysis of a hull girder using an equivalent beam representation is given in Appendix D. The input file (PREFX.HVCR) is given in Appendix E.

6.2 Update and Testing of Rudder Programs

The three programs associated with the rudder analysis capability in HVAST are RUDD1, RUDD2 and RUDD3. The purpose of RUDD1 is to generate the finite element model. RUDD2 provides the user with a means of automatically generating pressure loads. The fluid added mass is generated by RUDD3.

The initial work carried out was to update RUDD1, RUDD2 and RUDD3 to be compatible with the latest version of VAST. VAST had been modified since the rudder programs were initially developed and these changes had to be incorporated into the source code for each program. A driver program was also developed to make the generation of data files easier. A new option was added to the load generation program (RUDD2) to generate a data file for hydrostatic pressure loads.

When the appropriate updates were incorporated, each program was tested to see if the data files generated were correct.

A finite element model of the DDH280 rudder was then created using RUDD1. A hydrostatic pressure load data file and a fluid added mass data file were also generated using, respectively, RUDD2 and RUDD3. Plots of the structural and fluid finite element models are shown in Figures 6.4 and 6.5

A static analysis and a natural frequency analysis of the DDH280 rudder were performed using VAST. Plots of stress and natural mode shape contours were obtained. Samples of these plots are shown in Figures 6.6, 6.7 and 6.8.

6.3 Update and Testing of GRID2D

The HVAST modelling program GRID2D was updated so that its operation resembles that of VASGEN. Data input was simplified as much as possible. The user may define straight sides by specifying two end nodes, and curved sides may be generated by specifying four nodes along the edge. If material properties are the same for all patches, they need be only specified once. All output files are of the same format as those generated by VASGEN, and therefore can be combined.

GRID2d was tested on a sample bulkhead. Patch designations for the bulkhead are shown in Figure 6.9. The finite element mesh created by GRID2D is shown in Figure 6.10.

TABLE 6.1

PROGRAM NAMING SCHEME FOR HVAST PROGRAMS

New	Old	New	Old
HVEB	(new)	HVLD12	TDCPP3
HV2D	SHIP2D	HVLMA	MASADD
HVSP1	GRID2D	HVAM1	HULLAM
HVSPL	PANEL1	HVAM2	HULAM1
HVSP3	RUDD1	HVAM3	RUDD3
HVSP4	SHAFT	HVSKC	BULKHD1
HVASP5	MASTD	HGLDS1	(same)
HVLD1	HLOAD1	HGLDS2	HPLT5
HVLD2	HLOAD2	HGLDS3	LDPLOT
HVLD3	HLOAD3	HGDIS	HPLT1
HVLD4	HLOAD4	HGSTR1	(same)
HVLD5	BLKHD2	HGSTR2	(same)
HVLD6	HULLD1	HGSTR3	HPLT2
HVLD7	LOAGT	HGSTR4	HPLT3
HVLD8	BLAST1	HGSTR5	HPLT6
HVLD9	SHOCK1	HGSTR6	BMPLOT
HVLD10	RUDD2	HGSTR7	HPLT7
HVLD11	MASTDV	HGLWD	HPLT5

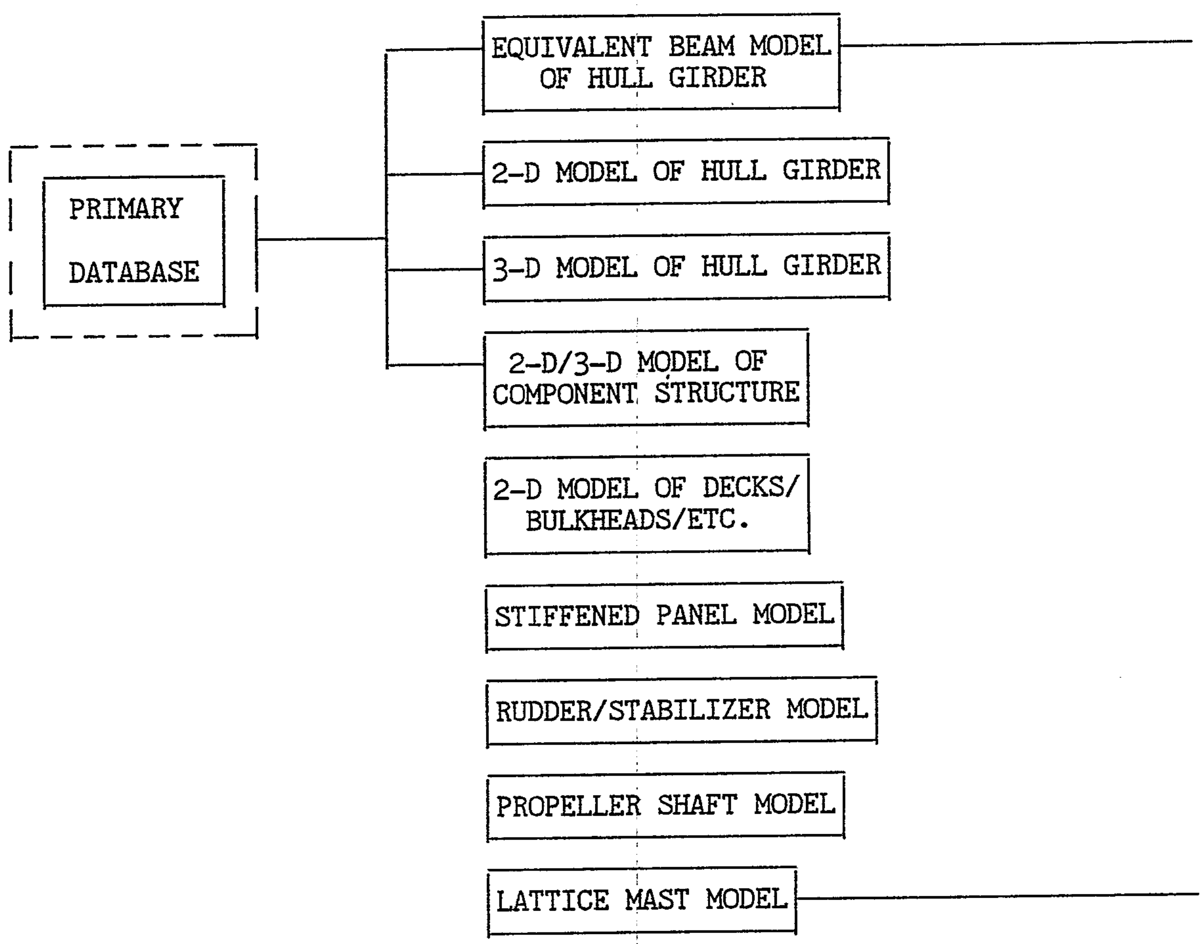


FIGURE 6.1: Model Types in HVAST Menu Driven System

ANALYSIS STEPS

1. Discretization of Structure into Finite Element Model
2. Specification of Skewed Coordinates/Multi-Point Constraints
3. Definition of Boundary Conditions
4. Addition of Lumped Masses
5. Hydrodynamic Added Mass
6. Still Water Loads
7. Hydrostatic/Hydrodynamic Pressure Loading
8. Wind/Air Blast/Underwater Explosion
9. Other Types of Loading
   - Loads from Edge Stress Distributions
10. Digitized Dynamic Loading
11. Inertial/Concentrated/Support Motion Loads
12. Special Modelling Options:
    A – Assembly of Individual Models as Substructures
    B – Top-down Modelling
    C – Bottom-up Modelling
    D – Mesh Refinement
13. Plot of Finite Element Models
14. Plot of Applied Loads
15. Other Types of Plots
    - Longitudinal Weight Distribution
    - Load Distribution at Specified Transverse Sections of Hull
16. Execution of External Programs
    A – VASTG/PATRAN (Plotting)
    B – VAST (Linear Analysis)
    C – ADINA (Non-linear Analysis)
    D – USA (Fluid-Structure Interaction)
17. Displacement Plotting
18. Stress Plotting

FIGURE 6.2: Analysis Steps in HVAST Menu Driven System

HVAST

APPLICATION MATRIX

Analysis Steps

1. Discretization of Structure into Finite Element Model
2. Specification of Skewed Coordinates/ Multi-Point Contraints
3. Definition of Boundary Conditions
4. Addition of Lumped Masses
5. Hydrodynamic Added Mass
6. Still Water Loads
7. Hydrostatic/Hydrodynamic Pressure Loading
8. Wind/Air Blast/Underwater Explosion
9. Other Types of Loading
   - Loads from Edge Stress Distributions
10. Digitized Dynamic Loading
11. Inertial/Concentration/Support Motion Loads
12. Special Modelling Options:
    - A – Assembly of Individual Models as Substructures
    - B – Top-Down Modelling
    - C – Bottom-Up Modelling
    - D – Mesh Refinement
13. Plot of Finite Element Models
14. Plot of Applied Loads
15. Other Types of Plots
    - Longitudinal Weight Distribution
    - Load Distribution at Specified Transverse Sections of Hull
16. Execution of External Programs
    - A – VASTG/PATRAN (Plotting)
    - B – VAST (Linear Analysis)
    - C – ADINA (Non-Linear Analysis)
    - D – USA (Fluid-Structure Interaction)
17. Displacement Plotting
18. Stress Plotting

Model Type	Analysis Step	Finite Element Modelling					Load Generation						Special Modelling Options	Input Plotting			External Programs	Output Plotting	
		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18
Equivalent Beam Model of Hull Girder	1	X		X	X	X			X		X	X		X	X	X	A B		X
2-D Model of Hull Girder	2	X		X	X	X					X	X		X	X	X	A B		
3-D Model of Hull Girder	3	X		X	X	X		X				X	A B C D	X	X	X	A B C D	X	X
2-D/3-D Models of Component Structures	4	X	X	X	X			X		X		X	A B C D	X	X		A B C D		X
2-D Model of Decks/Bulkheads/ Etc.	5	X	X	X	X					X	X	X	A D	X	X		A B C		X
Stiffened Panel Models	6	X		X	X				X	X	X	X		X	X		A B		X
Rudder/Stabilizer Models	7	X		X	X	X		X		X	X	X		X	X		A B C		
Propeller Shaft Model	8	X		X	X							X	A	X	X		A B		X
Lattice Mast Models	9	X		X	X				X		X	X		X	X		A B		

FIGURE 6.3: HVAST Application Matrix

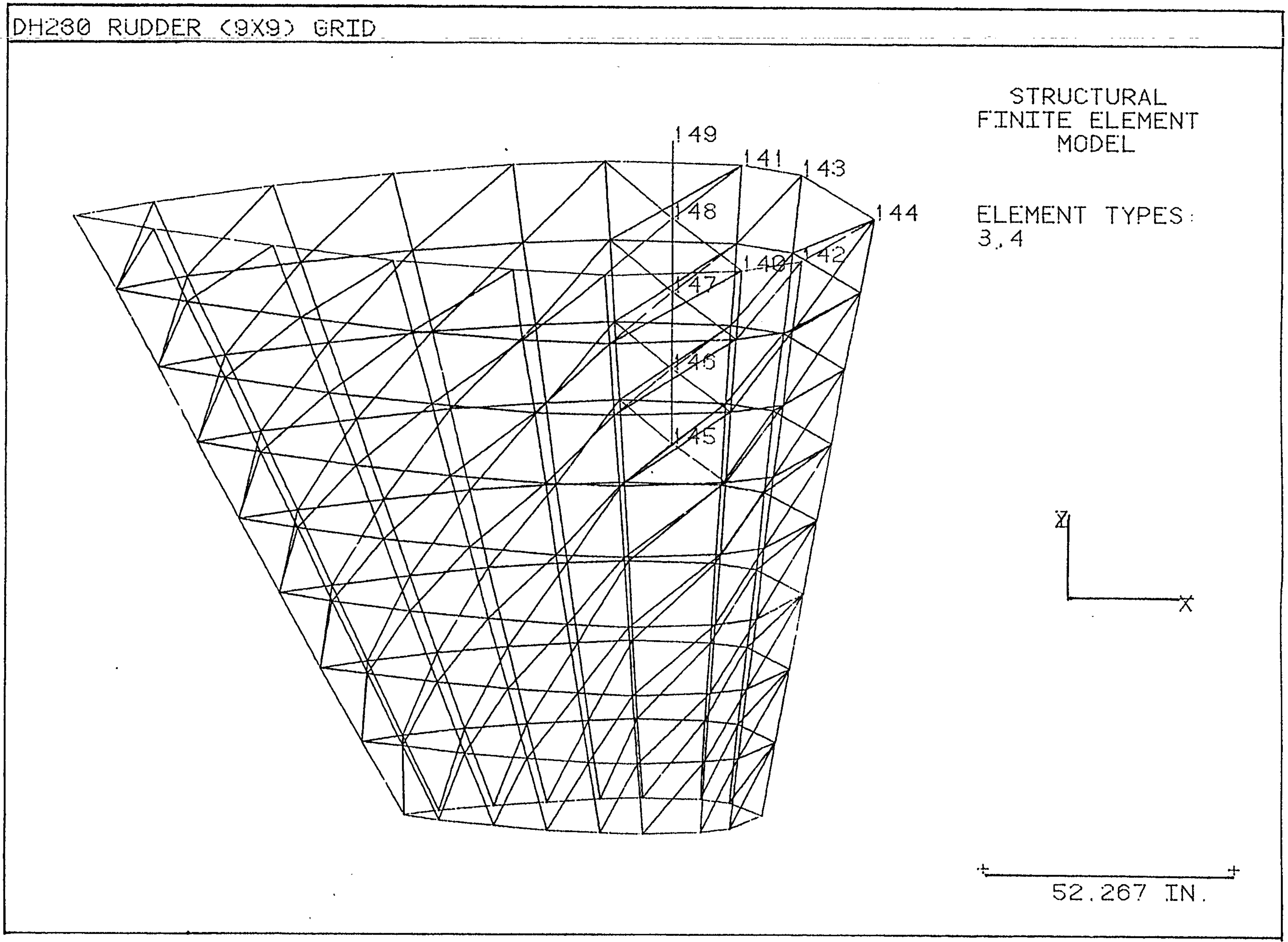


FIGURE 6.4: Structural Finite Element Model for Rudder

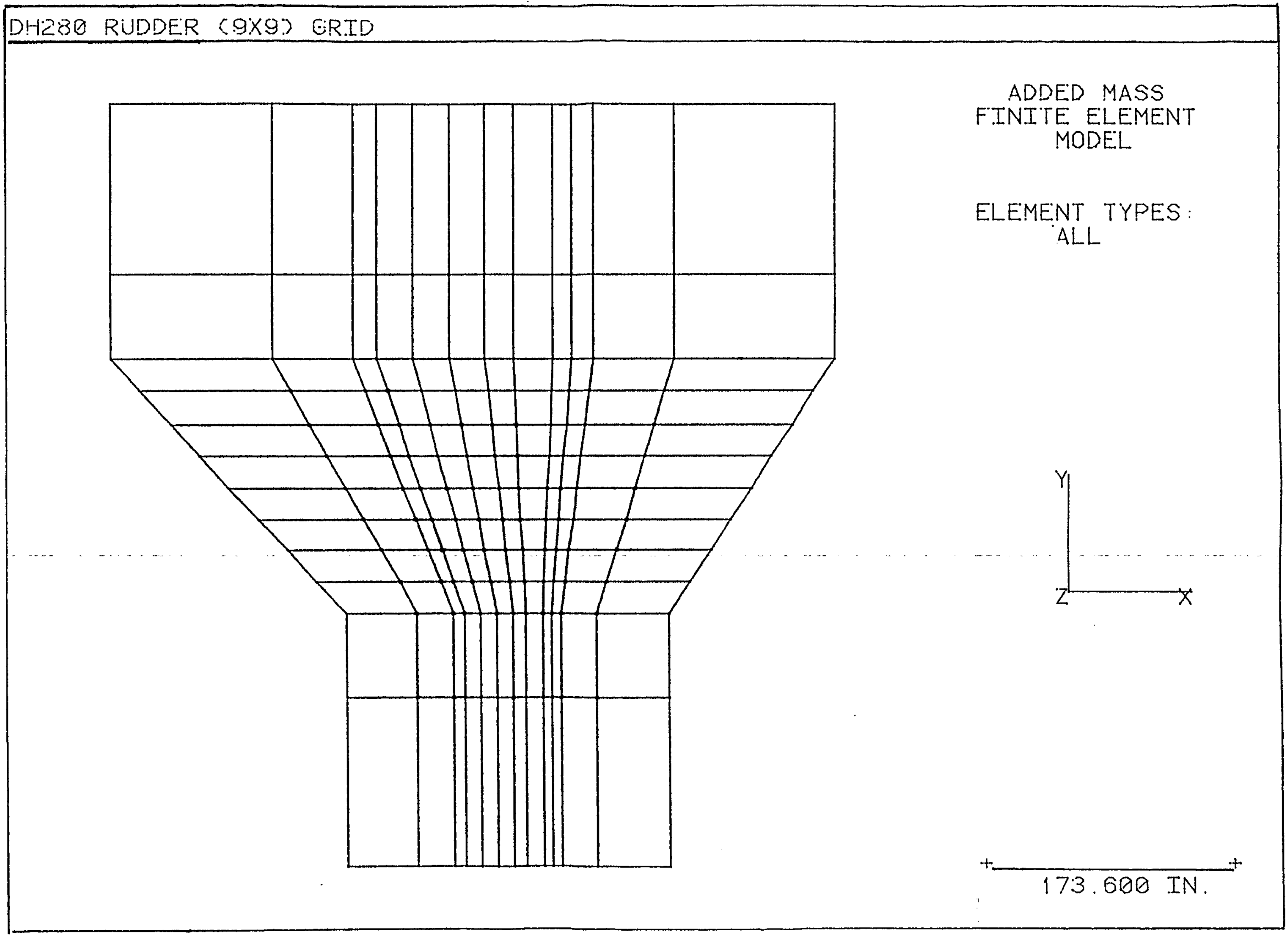
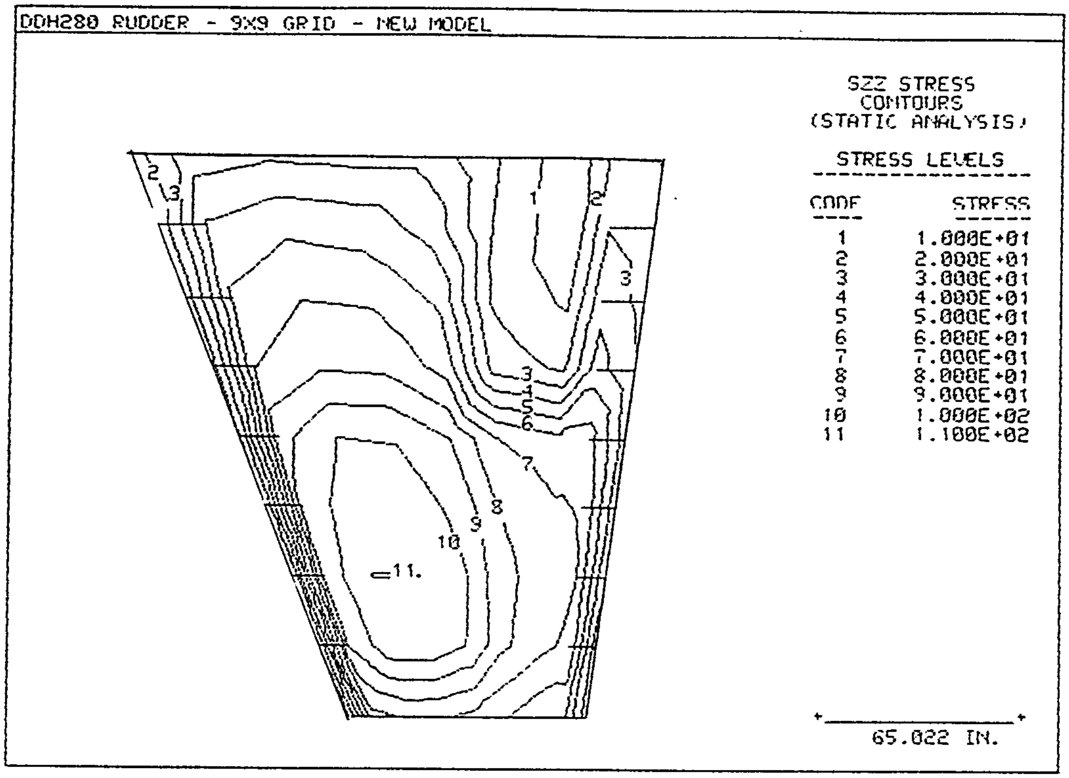


FIGURE 6.5: Fluid Fintie Element Model for Rudder



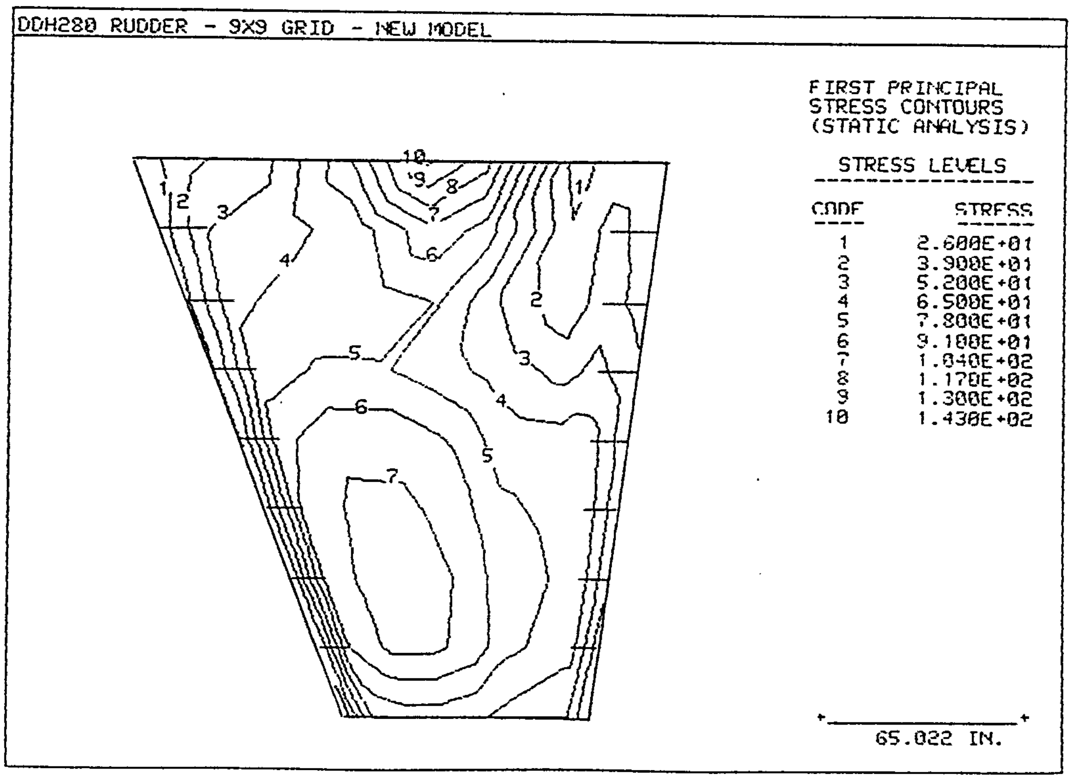
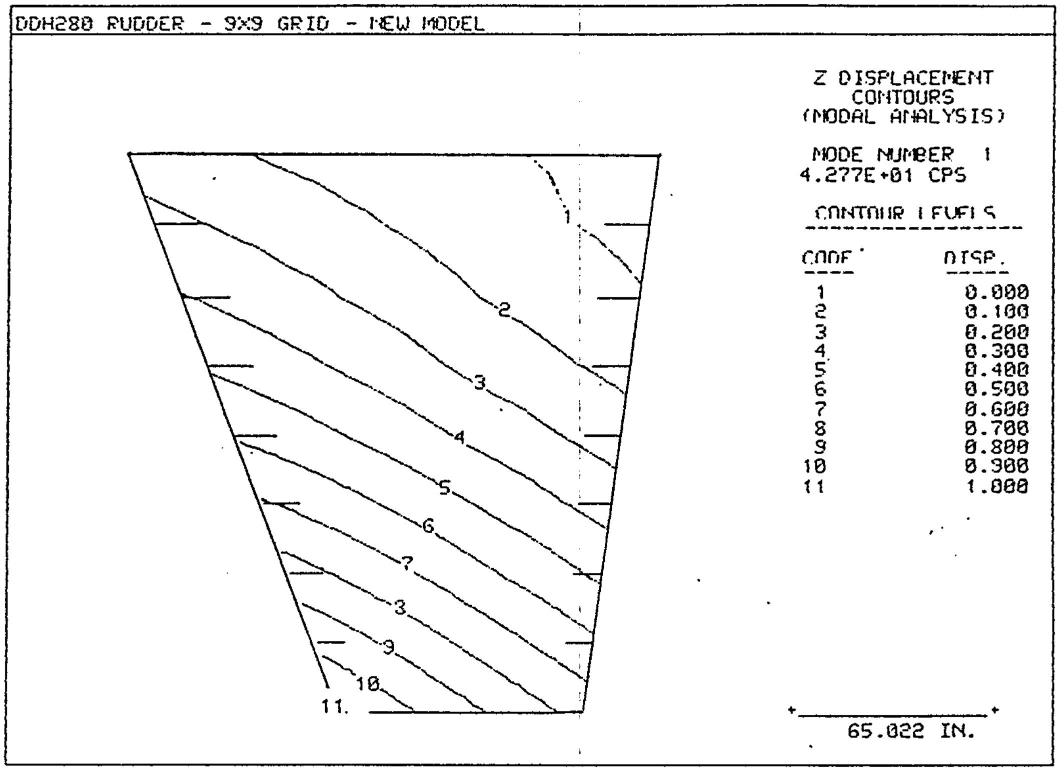


FIGURE 6.6: Stress Contour for Hydrostatic Pressure Loading of Rudder



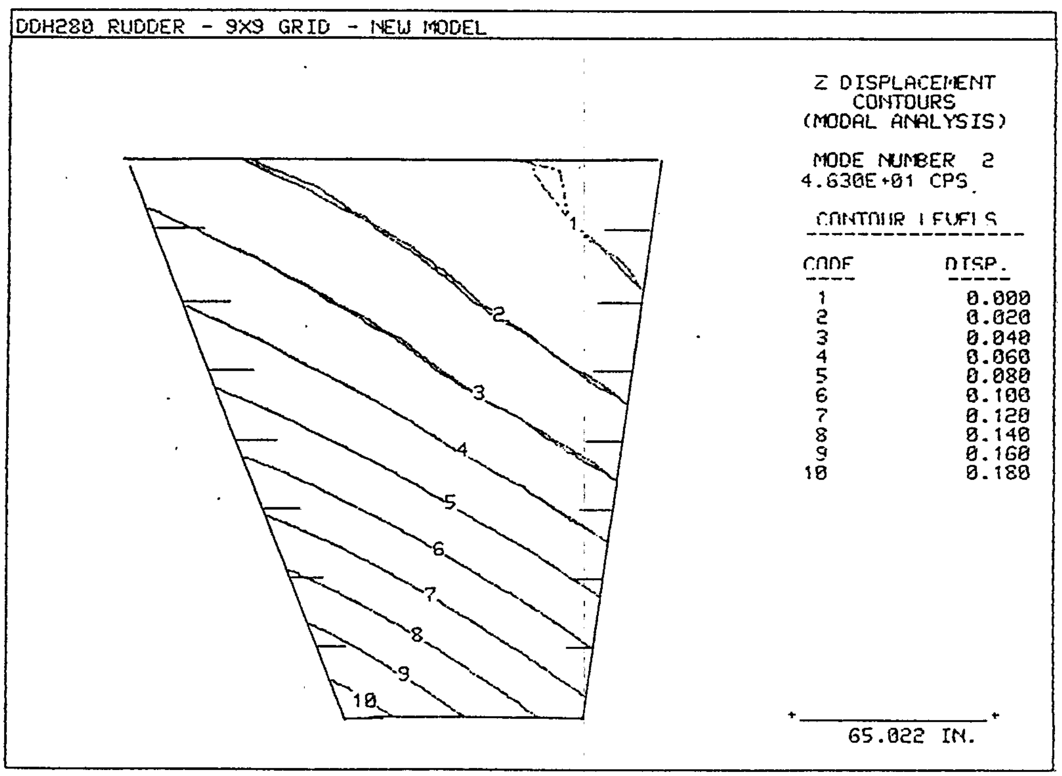
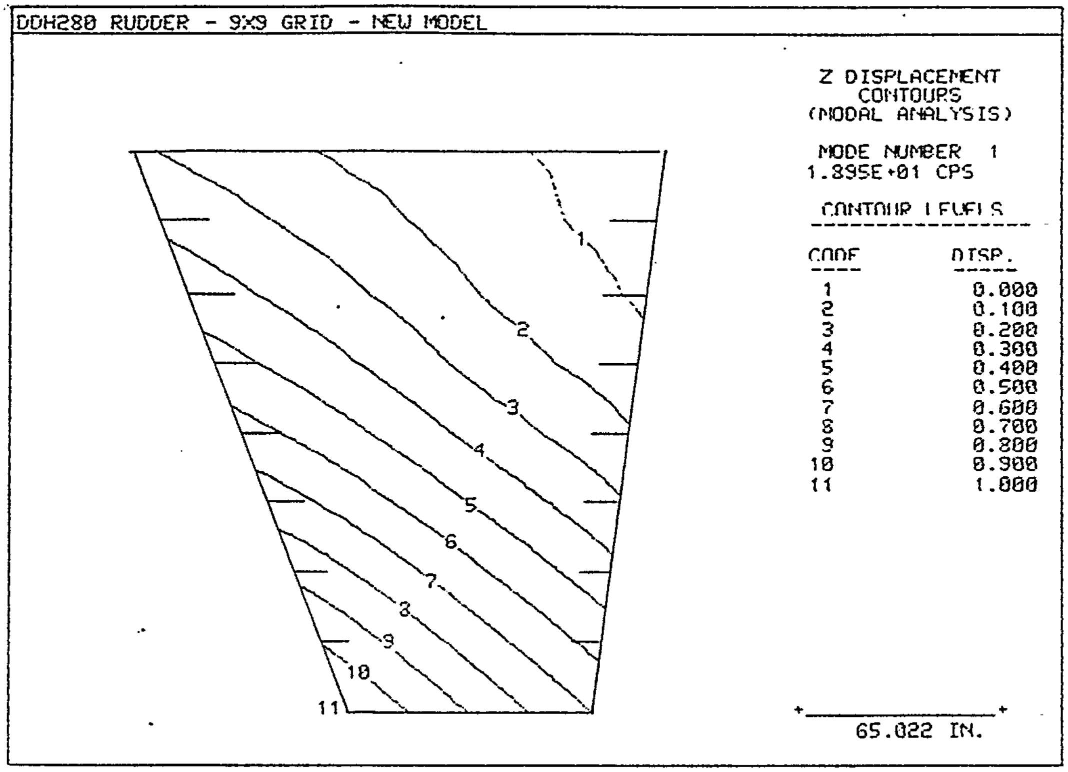


FIGURE 6.7: Natural Vibration Modes of Rudder in Air



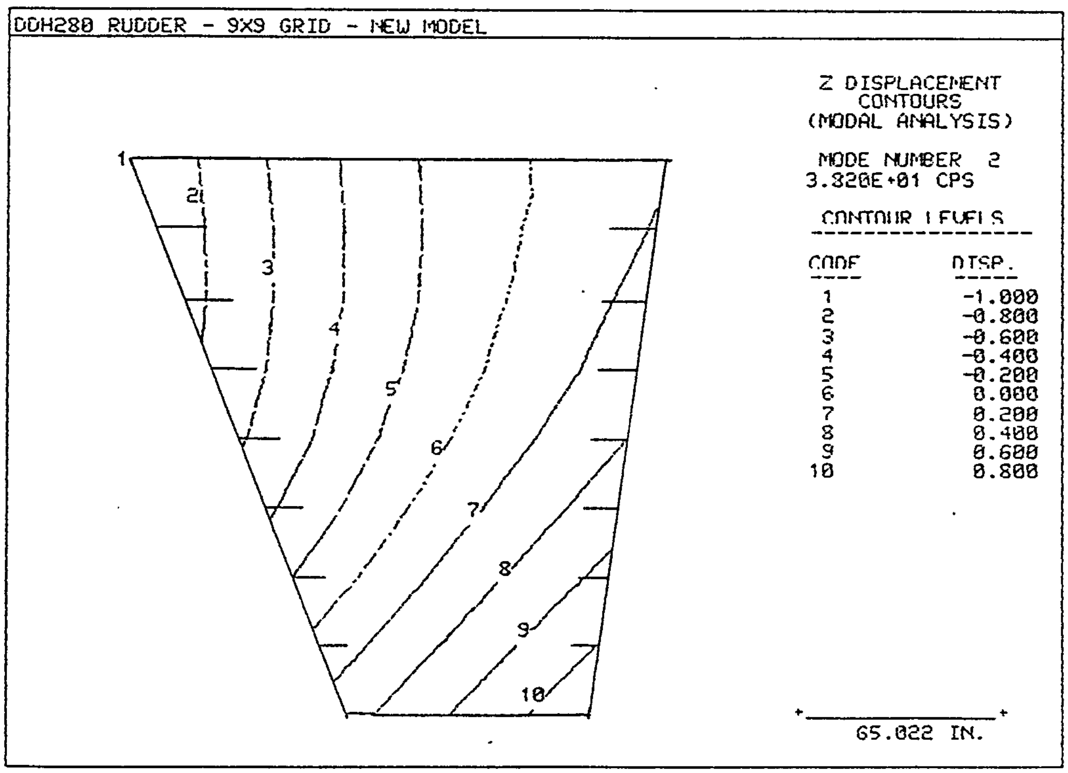


FIGURE 6.8: Natural Vibration Modes of Rudder in Water

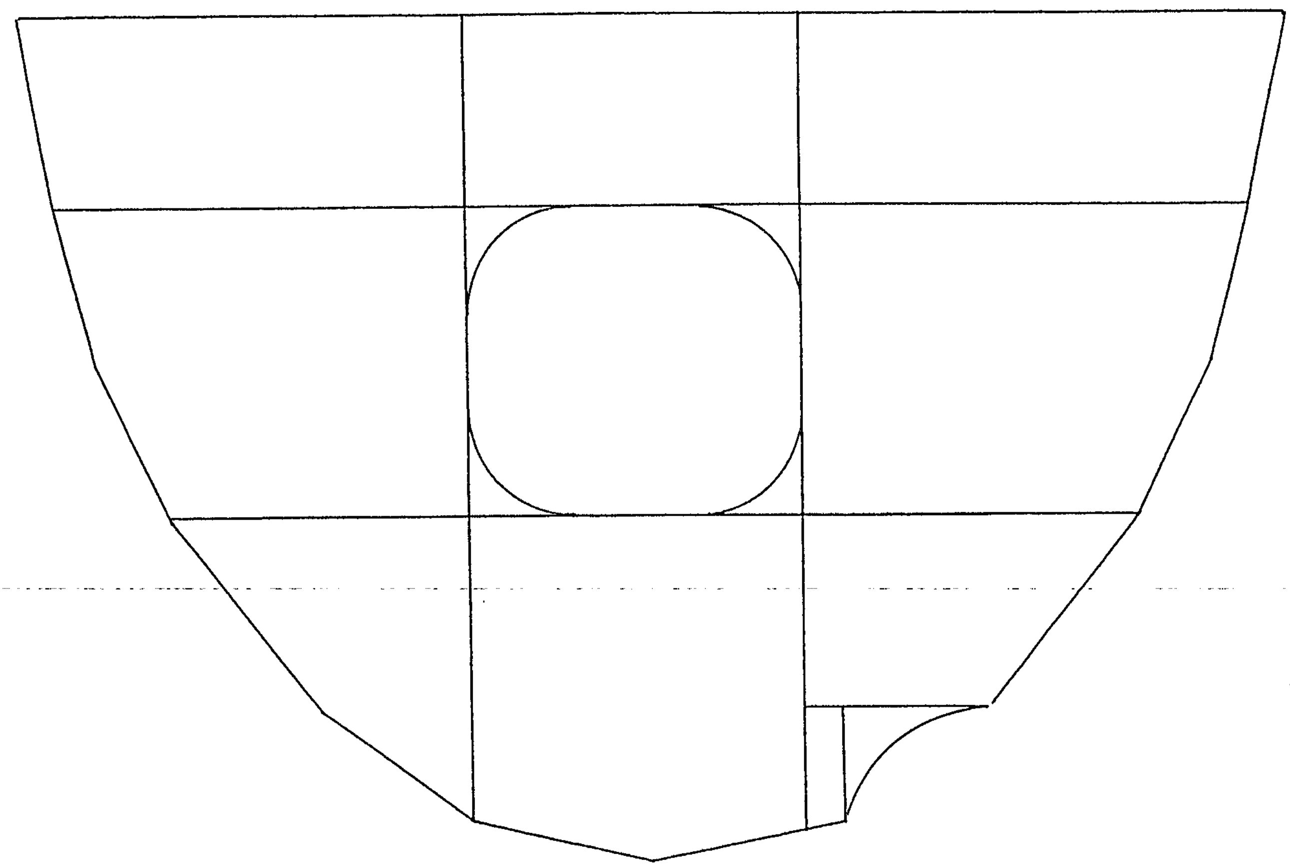


FIGURE 6.9: Sample Bulkhead - Patch Designation for GRID2D

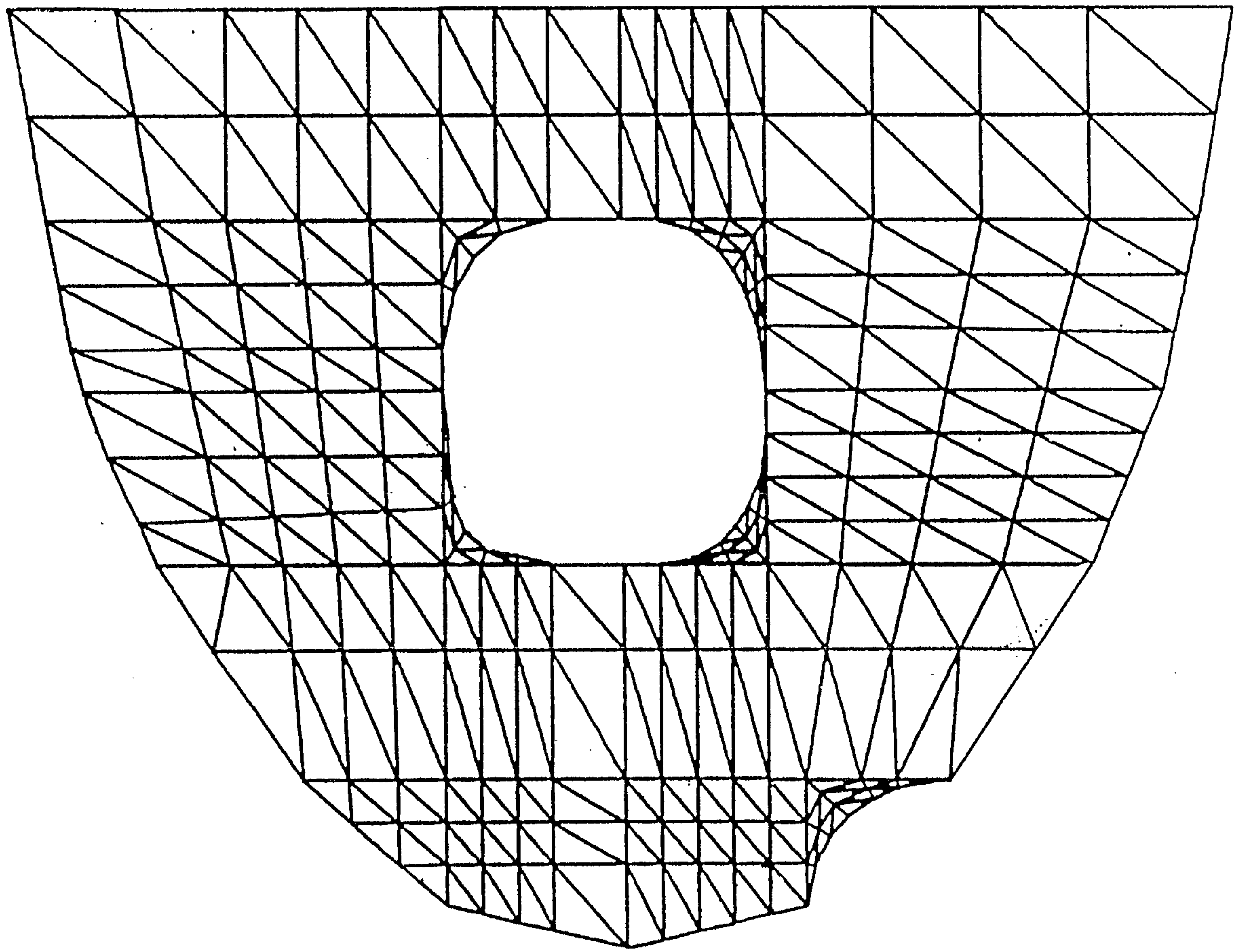


FIGURE 6.10: Sample Bulkhead - Model Generated by GRID2D

7.0 ASSEMBLY OF A STRUCTURE COMPOSED OF PREVIOUSLY GENERATED STRUCTURES

7.1 Equivalencing of Nodes

A program called UNITE was developed for the purpose of equivalencing nodes with the same geometric coordinates. The user has the option to make the equivalenced nodes master nodes, and create superelements of previously generated structures.

In cases such as lattice masts and lattice columns where the individual structural components may be made up of beams and bars, equivalencing of the shared beams and bars on the boundaries must be accounted for.

A detailed description of UNITE including operating instructions is provided in a separate report (Reference 1).

7.2 Master Node Generation

A program called MNGEN was developed for the purpose of adding additional master nodes to an assemblage of previously generated structures. Through the use of the terminal cursor, the user can identify master nodes in addition to those resulting from equivalencing using UNITE (see Section 7.1). The program can also be used to define, via cursor, boundary condition data, lumped mass data and multi-point constraint equations.

A detailed description of MNGEN including operating instructions is provided in a separate report (Reference 2).

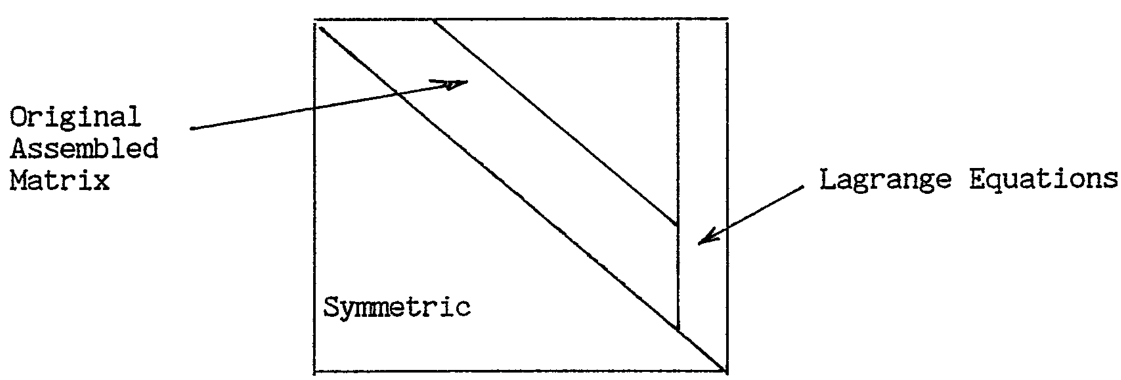
8.0 ARBITRARY BASE MOTION

Under a previous contract (Reference 6), a program called LOAD3 was developed for applying arbitrary base motion. This program was designed as a separate module for VAST. Under the current contract, this module was integrated with the module DISP3. The user can now apply arbitrary displacement, acceleration and velocity time histories for any number of degrees of freedom in the structure. Any general type of loading can be applied at the same time. The prescribed displacements, acceleration and velocities can be supplied on the PREFX.USE file or a separate file called PREFX.DIS.

9.0 IMPROVEMENTS IN MULTI-POINT CONSTRAINT ALGORITHM IN VAST

Under this task, an attempt was made to reduce running times in VAST when using multi-point constraints. The present multi-point constraint algorithm in VAST applies multi-point constraints by performing a series of transformations on the fully assembled stiffness and mass matrices. Under this contract, the algorithm was modified to carry out the appropriate matrix modifications using a random access file, instead of a sequential access file. Reductions in running times of a factor of 2 were found. However for problems with a large number of multi-point constraints, the modified algorithm still too seriously affects the analysis run time. An additional reduction in running times could be achieved if the present algorithm was changed to operate on the assembled matrices in block storage form, instead of the row storage form used now. However, a system whereby multi-point constraints are applied at the element matrix level, just prior to assembly, appears to be more appropriate.

The use of Langrange multipliers for applying multi-point constraints to the assembled global matrices in VAST was investigated. When using this method, an assembled global matrix is modified as shown below.



Characteristic of the method is the addition of a set of equations equal in number of the number of constraint equations. One very important fact is that the terms in the original assembled matrix remain unchanged. Consequently, in comparison to the existing multi-point constraint procedure in VAST, the stiffness/mass modification steps in an analysis is very fast. However, the bandwidth of the modified system becomes extremely large in comparison to the original. It will be noted, however, that this problem can be overcome considerably by using a profile solver.

Under this contract, the use of Lagrange multipliers in both static and eigenvalue analysis was investigated. New version of modules STIFM, MASSM, DECOM1, DISP1 and EIGEN1 had to be created for this purpose. Fairly major changes to these modules were required in order to accommodate the additional and unique 'Lagrangian' degrees-of-freedoms. Similar changes would be required for all other VAST modules.

A static analysis using the modified STIFM, DECOM1 and DISP1 modules was successful. However, running times were large due to the large bandwidth resulting from the Lagrange equations. The attempt at performing an eigenvalue analysis was unsuccessful due to problems with the new EIGEN1 module. Time did not permit further investigation.

This investigation has shown that it is possible to use Lagrange multipliers in VAST for applying multi-point constraints. However, this would require the use of a profile storage scheme for assembled stiffness and mass matrices, appropriate modifications to <u>all</u> VAST modules, and extensive testing.

10.0 TEST AND EVALUATION OF VASDAT

The purpose of this task was to test and evaluate the program VASDAT for all VAST04 analysis options, and make recommendations for an improved version of the program.

10.1 Basic Philosophy

VASDAT is an interactive program which was written to eliminate the need to refer to the VAST manual when creating input files for VAST. VASDAT was originally developed for the inexperienced user who would have no knowledge of the format of the VAST input files. This novice user would have the capability to sit at a terminal and at one session create all of the required input files for VAST.

This is a sound philosophy which should be adhered to in the development of VASDAT. However, because of developments in the area of preprocessing programs (eg. VASGEN) the capability exists to generate the geometry of a finite element model using other model generators. These other programs require no knowledge of the format of the VAST input files making the options to define geometry interactively in VASDAT somewhat redundant. Any efforts to improve geometry generation should be concentrated in these special preprocessing programs rather than in VASDAT. The options presently available in VASDAT represent the options which are available in the VAST user's manual whereas improved and more user friendly options are available in VASGEN or similar model generation programs.

It is our opinion that VASDAT should assume that a geometry file exists. If this is not acceptable, one option that could be feasible, if a GOM file does not exist, would be to call VASGEN from within VASDAT. This would be transparent to the user and would result in a program which could be used to create all required input files for any type of

analysis. In either case, VASDAT should be organized such that VASGEN could be easily incorporated at a later date.

The user should still have the option to modify the existing geometry data from within VASDAT. This could be used to add nodes, delete nodes, modify nodal coordinates, delete elements, add elements, etc. This would require the retention of the line by line editor which is presently available in VASDAT. More details on editing has been included in Section 10.2.

Although VASDAT should provide a novice user with the capability to run VAST without any knowledge of the format of the input files, VASDAT should also provide a tool for a sophisticated VAST user to make changes to existing input files. The novice user should not be burdened with cumbersome prompts requiring knowledge of VAST, however, the experienced user requires control over analysis options. This control could be established by the experienced user through master control codes.

It is recommended that the automatic restart capability should be eliminated from VASDAT. The novice user will be prompted for analysis type, etc., and will input any required information necessary to perform the analysis. VASDAT will activate all master control codes required for the analysis and no restart will be allowed. If however, an experienced user wishes to perform, for example, a new type of analysis from existing input files, the capability should exist for restarting from existing files (ex. geometry file). This is proposed by using a prompt asking whether or not control over master control codes is required. The prompt could read as follows:

```
IS CONTROL OVER MASTER CONTROL CODES REQUIRED:
         1= YES - recommended for experienced users only
         0= NO
```

If the user elects to control the generation of master control codes, the remaining execution of VASDAT will be determined by the selection of the master control codes. For example, if the user has previously performed a static analysis and wishes to change one boundary condition and to repeat the analysis, the following master control codes would be activated (by the user):

IELEMS = 0
IBANRD = 0
IASSEM = 0
ISTIFM = 1
IMASSM = 0
IDECOM = 1
IEIGEN = 0
ILOADS = 0
IDISPS = 1
ISTRES = 1

VASDAT will determine processing options based on the selection of these codes so that processing will start at the stiffness modifications section, will bypass load generation and continue for displacement and stress processing.

Processing data in this manner allows the versatility required to make the program effective for an experienced user while retaining the automation required by the novice. By recommending that a response of NO be input by the inexperienced user, VASDAT eliminates any prompts requiring a knowledge of the VAST input file format (i.e. Master Control Codes) for the novice.

10.2 Editing Data

The current editor in VASDAT requires a detailed knowledge of the format of the data input files. The editor works on the completed data files and allows the user to modify selected lines. A better approach would be to ask the user questions about what is to be changed. For example, if the boundary condition/stiffness modification section is to be modified, the user could be prompted as follows:

```
    SELECT THE APPROPRIATE PORTION OF THE STIFFNESS MODIFICATIONS
    SECTION TO BE ALTERED (OR CREATED, ETC.):
                0= NO CHANGES REQUIRED.
                1= ALTER SPRING-TYPE STIFFNESSES (I.E. NODAL BOUNDARY
                     CONDITIONS)
                2= ALTER EXTERNALLY GENERATED STIFFNESS MATRICES
                3= ALTER MULTI-POINT CONSTRAINT DATA
1

    SELECT ONE OF THE FOLLOWING OPTIONS:
                0= NO CHANGES
                1= CHANGE THE NUMBER OF BC'S (I.E. ADD OR DELETE NODES)
                2= ALTER BOUNDARY CONDITIONS FOR SPECIFIC NODES
2

    ENTER THE NUMBER OF BC'S TO BE ALTERED:
3

    ENTER NODE NUMBER 1 OF THE 3 BOUNDARY CONDITIONS TO BE MODIFIED:
    .
    .
    .
    ETC.
```

Because of the complexity of the input, this approach would be impractical for editing the geometry and load sections. It is therefore recommended that the line by line editor be retained to allow modification of these sections. In the remaining sections (i.e. bandwidth, reduction, assembly, stiffness modifications, decomposition, eigenvalue analysis, displacements and stresses) the above approach should be used which eliminates the requirement to have a detailed knowledge of the format of the VAST input files. The line by line editor could be retained in all sections as an additional feature.

10.3 Other Desirable Features

After any prompt in the program the user should be able to respond with an "S". The program will then terminate execution, save all responses input to this point, and give the option upon restart to continue the run. This would be accomplished through the creation of a session file onto which all interactive responses will be saved (with the exception of those input to execute graphics options).

The user will have the option to manually edit the session file if required, however the primary objective of the session file is to provide the capability to continue an interupted session. When VASDAT is rerun, if a session file exists, the user will be given the option either to start from scratch or to continue the run from the end of the session file.

During execution of a particular section of the program (eg. generating boundary conditions interactively) the user should have the option to ABORT. Control should then be transferred back to the beginning of that section and execution continued from that point. No other information, input prior to the current section, would be lost. Suggested input would be an "A".

10.4 Recommended Approach

The most efficient approach to providing the recommended options in VASDAT, would be to create a second generation program called VSDAT2. This program would be structured differently from the present version of VASDAT however, large sections of code would remain essentially unchanged. The main program would follow a more structured approach where each section is run sequentially and most data processing will be performed in separate subroutines. This approach will allow incorporation of the restart capability and provide a program which will be much easier to modify to include future options.

As a long term objective, the approach that should be followed would be to structure the program such that separate program modules could be incorporated as they become available. For example if a program is developed to allow graphical modification of boundary conditions, this capability should be easily incorporated into VSDAT2. This would be seen by the user as simply an additional option.

It is the feeling of the author that this is an approach which will make VSDAT2 a useful program which is unlikely to become obsolete as improved preprocessing options become available. The estimated labour requirements to update VASDAT would be approximately 8 to 10 weeks.

10.5 Sample Run of Proposed Program

A mock-up of two sample runs has been created (Appendix F) to demonstrate the ease with which an analysis could be performed. In the first sample run, an inexperienced user is required to set up a USE file to perform a linear static analysis. For the sake of simplifying required input, the scenario has been selected where all of the input files for VAST have been created previously. In a more realistic run, boundary condition data and concentrated load data would not likely exist

and would be prompted for by VASDAT. One comment which should be noted is the minimal input which is required to set up the USE file. Prompts pertaining to the solution detail in VAST (for example decomposition, assembly data, etc.) have all been eliminated for the novice user.

The second sample run is one which might be performed by an experienced VAST user who wishes to restart an analysis. It is again assumed that all VAST input files exist. In this run the user wishes to restart the analysis from the assembly section (i.e. the geometry section has been run and bandwidth reduction is not required).

11.0 DIGITIZING DYNAMIC LOAD TIME HISTORIES

The ability to generate a time history of loading by digitizing a load versus time curve was developed. A special program called LOADGT was developed for this purpose. The load history plot is placed on the digitizer tablet, appropriately scaled and then converted to a VAST PREFX.LOD file. Multiple loading functions can be processed. Each loading function is digitized according to node and degree-of-freedom, which are inputted via the terminal keyboard. A description of the program, its capabilities and operating instructions are given in Appendix G.

12.0 COMPUTER SYSTEMS

All programs developed under this contract were developed on the DEC-20 system. They were later converted to run on the VAX-750. The actual running and testing of HVAST and its suite of programs was conducted on the VAX-750. Not all HVAST options could be tested and evaluated under this contract.

The programs VASTBC, UNITE and MNGEN are operational on both DEC-20 and VAX-750 systems.

REFERENCES

1. E.J. CROCKER and M.W CHERNUKA. Program UNITE to Assemble Substructured Finite Element Models - Progress Report, Martec Limited, Halifax, N.S., August 1987.

2. E.J. CROCKER and M.W CHERNUKA. Program MNGEN for Interactive Generation of Auxiliary Master Nodes and Optional Input of Associated Finite Element Data - Progress Report, Martec Limited, Halifax, N.S., August 1987.

3. O.F. HUGHES. Ship Structural Design: A Rationally-Based, Computer-Aided Optimization Approach, John Wiley and Sons, 1983.

4. S. ANDO. POSBOW-FORTRAN Program for Calculating Pressures on Ship Hulls Balanced on Standard Wave, DREA Research Note H/83/3, November 1983.

5. Boundary Conditions File Generation Program for VAST - VASTBC User's Manual, Martec Limited, Halifax, N.S., November 1989.

6. M.E. NORWOOD, M.W. CHERNUKA, G.L. NIGH, D.F. FANNING and R.P. JATEGAONKAR. Strength and Vibration Analysis of Naval Ships Hulls and Appendages - Final Report, Martec Limited, Halifax, N.S., September 1984.

7. N.I. MUSKHELISHVILI. Some Basic Problems of the Mathematical Theory of Elasticity, P. Noordhoff Ltd., Groningen (1953).

8. S.G. MIKLIN. Integral Equations, Pergamon Press, New York (1957).

9. O.D. KUPRADZE. Potential Methods in the Theory of Elasticity, Daniel Davey & Co., New York (1965).

10. V.J. SMIRNOV. Integral Equations and Partial Differential Equations, In "A Course in Higher Mathematics", Volume IV, Addison-Wesley (1964).

11. M.A. JASWON. Integral equation methods in potential theory - Part I. Proc. R. Soc., Ser. A, Vol. 275, pp. 23-32 (1963).

12. G.T. SYMM. Integral equation methods in potential theory - Part II. Proc. R. Soc., Ser. A, Vol. 275, pp. 33-46 (1963).

13. C.E. MASSONNET. Numerical use of integral procedures in stress analysis, In "Stress Analysis", Ed. by O.C. ZIENKIEWICZ and G.S. HOLISTER, Wiley (1966).

14. J.L. HESS and A.M.O. SMITH. Calculation of potential flow about arbitrary bodies. In "Progress in Aeronautical Sciences", Vol. 8, Ed. by D. KUCHEMANN. Pergamon Press (1967).

15. T.A. CRUSE and F.J. RIZZO. A direct formulation and numerical solution of the general transient elasto-dynamic problem. Journal of Mathematical Analysis and Applications, Vol. 22, pp. 244-259 (1968).

16. F.J. RIZZO. An integral equation approach to boundary value problems of classical elastostatics. Quarterly of Applied Mathematics, Vol. 25, pp. 83-95 (1967).

17. T.A. CRUSE. Numerical solutions in three-dimensional elastostatics. Int. J. Solids Struct., Vol. 5, pp. 1259-1274 (1969).

18. F.J. RIZZO and D.J. SHIPPY. An application of the correspondence principle of linear viscoelasticity theory. Society of Industrial and Applied Mathematics, Journal of Applied Mathematics, Vol. 21, pp. 321-330 (1971).

19. F.J. RIZZO and D.J. SHIPPY. An advanced boundary integral equation method for three-dimensional thermoelasticity. Int. J. Numer. Methods in Engng., Vol. 11, pp. 1753-1768 (1977).

20. T.A. CRUSE and W. VAN BUREN. Three-dimensional elastic stress analysis of a fracture specimen with an edge crack. Int. J. Fracture Mech., Vol. 7, pp. 1-16 (1971).

21. T.A. CRUSE. Numerical evaluation of elastic stress intensity factors by the boundary integral equation method, pp. 153-170. In "The Surface Crack: Physical Problems and Computational Solutions", Ed. by J.L. SWEDLOW, ASME, New York (1972).

22. M.D. SNYDER and T.A. CRUSE. Boundary integral equation analysis of cracked anisotropic plates. Int. J. Fracture, Vol. 11, pp. 315-328 (1975).

23. T.A. CRUSE. Boundary Integral Equation Method for Three-Dimensional Elastic Fracture Mechanics Analysis. Air Force Office of Scientific Research. TR-75-0813, Accession No. ADA 011660 (1975).

24. T.A. CRUSE and G.J. MEYERS. Three-dimensional fracture mechanics analysis. Proc. ASCE, Journal of Structural Division, Vol. 103, pp. 309-320 (1977).

25. M.A. JASWON and A.R. PONTER. An integral equation solution of the torsion problem. Proceedings of the Royal Society, London, Series A, Vol. 273, pp. 237-246 (1963).

26. M.A. JASWON and M. MAITI. An integral equation formulation of plate bending problems. Journal of Engineering Mathematics, Vol. 2, pp. 83-93 (1968).

27. D.A. NEWTON and H. TOTTENHAM. Boundary value problems in thin shallow shells of arbitrary plan form. Journal of Engineering Mathematics, Vol. 2, pp. 211-224 (1968).

28. T.A. CRUSE and F.J. RIZZO. A direct formulation and numerical solution of the general transient elastodynamic problem - Part I, Journal of Mathematical Analysis and Applications, Vol. 22, pp. 244-259 (1968).

29. T.A. CRUSE. A direct formulation and numerical solution of the general transient elastodynamic problem - Part II. Journal of Mathematical Analysis and Applications, Vol. 22, pp. 341-355 (1968).

30. R.P. SHAW. Retarded potential approach to the scattering of elastic waves by rigid obstacles of arbitrary shape. Journal of the Acoustical Society of America, Vol. 44, pp. 745-748 (1968).

31. J.L. SWEDLOW and T.A. CRUSE. Formulation of boundary integral equations for three-dimensional elasto-plastic flow. Int. J. Solids Struct., Vol. 7, pp. 1673-1681 (1971).

32. A. MENDELSON. Boundary Integral Methods in Elasticity and Plasticity, NASA TN-D7418 (1973).

33. A. MENDELSON and L.U. ALBERS. Application of boundary integral equations to elasto-plastic problems. In "Boundary Integral Equation Method: Computational Applications in Applied Mechanics". Ed. by T.A. CRUSE and F.J. RIZZO. ASME Proceedings AMD-11 (1975).

34. P. RICCARDELLA. An Implementation of the Boundary Integral Technique for Plane Problems of Elasticity and Elasto-Plasticity. PhD Thesis, Carnegie Mellon University, Pittsburg, PA (1973).

35. T.A. CRUSE. An improved boundary integral equation method for three-dimensional elastic stress analysis. Computers and Structures, Vol. 4, pp. 741-754 (1974).

36. P.K. BANERJEE and G.C.W. MUSTOE. The boundary element method for two-dimensional problems of elasto-plasticity, pp. 283-300. In "Recent Advances in Boundary Element Methods", Ed. by C.A. BREBBIA, Pentech Press, Plymouth, Devon, UK (1978).

37. V. KUMAR and S. MUKHERJEE. A boundary-integral equation formulation for time-dependent inelastic deformation in metals. Int. J. Mechanical Sciences, Vol. 19, pp. 713-724 (1977).

38. S. MUKHERJEE and V. KUMAR. Numerical analysis of time-dependent inelastic deformation in metallic media using the boundary-integral equation method. ASME Journal of Applied Mechanics, Vol. 45, pp. 785-790 (1978).

39. S. MUKHERJEE and M. MORJARIA. Comparison of boundary element and finite element methods in the inelastic torsion of prismatic shafts. Int. J. Numer. Methods Engng., Vol. 17, pp. 1576-1588 (1981).

40. M. MORJARIA and S. MUKHERJEE. Inelastic analysis of transverse deflection of plates by the boundary element method. ASME Journal of Applied Mechanics, Vol. 47, pp. 291-296 (1980).

41. M. MORJARIA and S. MUKHERJEE. Numerical analysis of planar, time dependent inelastic deformation of plates with cracks by the boundary element method. Int. J. Solids Struct., Vol. 17, pp. 127-143 (1981).

42. J.M. BOISSENOT, J.C. LACHAT and J.O. WATSON. Etude par equations integrales d'une eprouvette. C.T.-15, Rev. Phys. Appl., Vol. 9, pp. 611-615 (1974).

43. J.C. LACHAT. A Further Development of the Boundary Integral Technique for Elastostatics, PhD thesis, University of Southampton (1975).

44. J.C. LACHAT and J.O. WATSON. A second generation boundary integral equation program for the three-dimensional elastic analysis. In "Boundary Integral Equation Method: Computational Applications in Applied Mechanics. Ed. by T.A. CRUSE and F.J. RIZZO., ASME Proceedings AMD-11 (1975).

45. J.C. LACHAT and J.O. WATSON. Effective numerical treatment of boundary integral equations. Int. J. Numer. Methods Engng., Vol. 10, pp. 991-1005 (1976).

46. T.A. CRUSE and R.B. WILSON. Advanced applications of boundary integral equation methods. Nuclear Engng. and Design, Vol. 46, pp. 223-234 (1978).

47. C.L. TAN and R.T. FENNER. Three-dimensional stress and analysis by the boundary integral equation method. J. Strain Analysis, Vol. 13, pp. 213-219 (1978).

48. C.L. TAN and R.T. FENNER. Elastic fracture mechanics by the boundary integral equation method. Proc. Roy. Soc. Lond., Vol. A369, pp. 243-269 (1979).

49. O.C. ZIENKIEWICZ, D.W. KELLY and P. BETTESS. The coupling of the finite element method and boundary solution procedures. Int. J. Numer. Methods in Engng., Vol. 11, pp. 355-375 (1977).

50. R.P. SHAW and W. FALBY. A combined finite element-boundary integral equation method. Proceedings of First Symp. on Innovative Numerical Analysis in Applied Engineering Science, Versailles (1977).

51. R.P. SHAW. Coupling boundary integral method to other numerical techniques, Int. Symp. Recent Advances in BEM, Southampton, England (July 1978).

52. J.R. OSIAS, R.B. WILSON and L.A. SEITELMAN. Combined boundary integral equation finite element analysis of solids. Proceedings of First Symp. on Innovative Numerical Analysis in Applied Engineering Science, Versailles (1977).

53. D.W. KELLY, G.G.W. MUSTOE and O.C. ZIENKIEWICZ. Coupling boundary element methods with other numerical methods, pp. 251-285, In "Developments in Boundary Element Methods - Vol. 1", Ed. by P.K. BANERJEE and R. BUTTERFIELD. Applied Science Publishers, London (1979).

54. C.A. BREBBIA and P. GEORGIOU. Combination of boundary and finite elements for elastostatics. Applied Math. Modelling, Vol. 3, pp. 212-220 (1979).

55. M. MARGULIES. Combination of the boundary element and finite element methods, pp. 259-288. In "Progress in Boundary Elements - Vol. 1", Ed. by C.A. BREBBIA, Pentech Press, Plymouth (1981).

56. L. BOLTEUS and O. TULLBERG. BEMSTAT - a new type of boundary element program for two-dimensional elasticity problems. In "Boundary Element Methods", Ed. by C.A. BREBBIA, Springer-Verlag, Berlin (1981).

57. G. BEER and J.L. MEEK. The coupling of boundary and finite element methods for infinite domain problems in elastoplasticity, pp. 575-591. In "Proceedings of Third Int. Seminar on Boundary Element Methods", Ed. by C.A. BREBBIA, Irvine (1981).

58. G. BEER. Finite element, boundary element and coupled analysis of unbounded problems in elastostatics. Int. J. Numer. Methods in Engng., Vol. 19, pp. 567-580 (1983).

59. F. HARTMANN. The derivation of stiffness matrices from integral equations. Applied Math. Modelling, Vol. 5, pp. 355-365 (1981).

60. O. TULLBERG and L. BOLTEUS. A critical study of different boundary element stiffness matrices, pp. 624-637. In "Proceedings of Fourth International Seminar on Boundary Element Methods in Engineering". Ed. by C.A. BREBBIA, Southampton, England, September 1982. Published by Springer-Verlag, Berlin (1982).

61. M.A. JASWON and M. MAITI. An integral equation formulation of plate bending problems. J. Eng. Math., Vol. 2, pp. 83-93 (1968).

62. M. MAITI and S.K. CHAKRABARTY. Integral solutions for simply supported polygonal plates. Int. J. Engng. Sci., Vol. 12, pp. 793-806 (1974).

63. N. ALTIERO and D.L. SIKARSKIE. A boundary integral method applied to plates of arbitrary plan form. Computers and Structures, Vol. 9, pp. 163-168 (1978).

64. B.C. WU and N.J. ALTIERO. A boundary integral method applied to plates of abitrary plan form and arbitrary boundary conditions. Computers and Structures, Vol. 10, pp. 703-707 (1979).

65. H. TOTTENHAM. The boundary element method for plates and shells, pp. 173-205. Chapter 8 in "Developments in Boundary Element Methods - I", Ed. by P.K. BANERJEE and R. BUTTERFIELD, Applied Science Publishers Ltd., London (1979).

66. E.B. HANSEN. Numerical solution of integro-differential and singular integral equations for plate bending problems, Journal of Elasticity, Vol. 6, pp. 39-56 (1976).

67. D.J. FORBES and A.R. ROBINSON. Numerical Analysis of Elastic Plates and Shallow Shells by an Integral Equation Method. Mechanics Research Series Report 345, The University of Illinois (1969).

68. G. BEZINE. Boundary integral formulation for plate flexure with arbitrary boundary conditions. Mechanics Research Communications, Vol. 5, pp. 197-296 (1978).

69. G. BEZINE and D. GAMBY. A new integral equation formulation for plate bending problems, pp. 327-342. In "Recent Advances in Boundary Element Methods", Ed. by C.A. BREBBIA, Pentech Press, London (1978).

70. M. STERN. A general boundary integral formulation for the numerical solution of plate bending problems. Int. J. Solids Struct., Vol. 15, pp. 769-782 (1979).

71. M. STERN. Boundary integral equations for bending of thin plates, pp. 158-181. In "Progress in Boundary Element Methods", Pentech Press, London (1983).

72. M. STERN and T.-L. LIN. Thin elastic plates in bending, pp. 91-119. Chapter 4 in "Developments in Boundary Element Methods - 4", Ed. by P.K. BANERJEE and J.O. WATSON. Elsevier Applied Science Publishers, London (1986).

73. R. BUTTERFIELD. New concepts illustrated by old problems, pp. 1-20. In "Developments in Boundary Element Methods -I", Ed. by P.K. BANERJEE and R. BUTTERFIELD. Applied Science Publishers, Ltd. (1979).

APPENDIX A

PROGRAM VASTBC

VASTBC

VASTBC plots structural finite element models from VAST input data files and allows the user to define new or to edit existing boundary condition data interactively using the graphics cursor. The operation of the program is very similar to the VAST-G program, PLOTV1, and hence offers the user much control over what is shown on the plot. The initial operation of VASTBC proceeds as PLOTV1 with the user specifying nodes to be plotted, elements to be plotted, viewing parameters and various other specifications. Boundary conditions may be indicated if they are contained on the data file or may be plotted after they are generated. VASTBC differs from PLOTV1 in the appearance of the plot and the operation of the program following the appearance of the plot. The right hand side of the plot is reserved for the display of the legend containing the instructions for the next step of operation. The current options available in VASTBC are shown in Figure A.1 and are activated from the graphics cursor and keyboard. A dialog area is reserved at the bottom of the plot for data entry and prompting.

Because VASTBC is plotting VAST input data which may exist in a number of different files, the input files required are not always the same. The structural data will be contained in either the PREFX.USE or PREFX.GOM files. In the case that there are no substructures/superelements, VASTBC will operate properly with only the PREFX.GOM file. However, in this case original boundary conditions, if present, cannot be plotted or modified. If there are substructures, superelements, or original boundary conditions, PREFX.USE must be present with the necessary data to plot these features.

The execution of VASTBC results in the creation of a PREFX.SMD file containing the boundary condition data as specified through interactive prompting for nodal points selected using the graphics cursor. The nodal points may be selected individually or multiply. If the model contains

substructures/superelements, then the application of a boundary condition to a node not specified as a master node will result in the superelement data being revised to reflect this change. Additionally, VASTBC allows the user to specify a skew coordinate system for a selected nodal point.

Operating Instructions for Program VASTBC

Files Required: PREFX.USE, PREFX.GOM, PREFX.SED, PREFX.SMD

Terminal Response:
(Note: all input is in free format unless otherwise specified.)

(1) PROGRAM VASTBC: VAST GRAPHICS PROGRAM FOR GENERATING/MODIFING BOUNDARY CONDITIONS FOR FINITE ELEMENT MODELS

(2) ENTER VAST FILE NAME PREFX

The user inputs the VAST file name 'PREFX'. This is the prefix name which identified the data file sets, and may be up to 5 characters long. VASTBC searches for the data files with the specified prefix name.

(3) WHAT IS THE LINE SPEED?

The user should enter the baud rate at which the terminal is communicating with the host computer.

(4) IDENTIFY TERMINAL TYPE ACCORDING TO RESOLUTION, CURSOR AND COLOUR CAPABILITY:

```
ENTER 0 FOR TEKTRONIX 4006 OR EQUAL (LOW RES/NO CURS/NO COL)
      1 FOR TEKTRONIX 401* OR EQUAL (LOW RES/CURSORS/NO COL)
      2 FOR TEKTRONIX 4014/15 (HI RES/CURSORS/NO COL)
      3 FOR TEKTRONIX 4113 OR EQUAL (COLOUR):
```

The user should enter a 1, 2 or 3 according to the type of features the terminal supports. The user should avoid responding with '3' as colour graphics have not yet been reliably incorporated into

VASTBC. VASTBC requires cursor capability and if a '0' is entered the program will terminate.

(5) IDENTIFY SPECIAL HARDWARE REQUIREMENTS:
ENTER 0 IF NO EXTRA EQUIPMENT IS TO BE USED
1 FOR TEKTRONIX 4907 FILE MANAGER
2 FOR TEKTRONIX 4662 PLOTTER
3 FOR TEKTRONIX 4663 PLOTTER
4 FOR CALCOMP 1012 PLOTTER

The user should enter 0 in response to this prompt.

(6) ENTER 0 FOR VIEWING VECTOR APPROACH
OR 1 FOR FINITE ANGULAR ROTATIONS:

The user defines which viewing approach is desired. See the section in the VASTG manual [1] on view specifications for a description of the two different approaches (prompts 21 and 22).

(7) ENTER 0 TO GENERATE NEW BOUNDARY CONDITION DATA
OR 1 TO MODIFY EXISTING BOUNDARY CONDITION DATA

This prompt is omitted if there is not any existing boundary condition data.

(8) ENTER 0 TO SPECIFY SKEW DATA VIA PROMPT
OR 1 TO SPECIFY SKEW DATA VIA CURSOR

This prompt indicates the method employed to determine the direction of the Y-axis in the skewed system for each node to be assigned a skew coordinate system. If '0' is entered then as nodes are selected for skew data the user is prompted for the rotation in degrees about the Z-axis. If '1' is entered the following message appears:

"TO SPECIFY A SKEW ON A NODE USING THE CURSOR SELECT THE NODAL POINT AND A POINT PERPENDICULAR TO THE SELECTED NODE."

Then a node is selected and a nodal point perpendicular to the node is selected to establish the skew data. The Y-axis of the skew coordinate system is calculated to pass through these two points, and is assumed positive in the direction from the nodal point to the second point.

Statements (1) to (7) appear only once, at the beginning of each execution of VASTBC. The parameters specified in response to these prompts may only be changed by restarting VASTBC. The remaining input specifications may be modified when running VASTBC and are divided into three sections, (1) PLOT specifications, (2) VIEW specifications, (3) CURSOR options.

(9) * PLOT SPECIFICATIONS *

(10) DO YOU WANT TO PLOT ANY NODES? (0=NO):

If the user wants any nodes identified on the plot, a non-zero response should be entered. If the user enters 0, prompts (11) to (15) are omitted.

(11) ENTER NUMBER OF SUPERELEMENT FOR NODE PLOTTING
(<0 FOR SAME OPTIONS AS PREVIOUS SUPERELEMENT
=0 TO TERMINATE NODE PLOTTING SPECIFICATIONS
>0 TO SPECIFY NODE PLOTTING OPTIONS):

This prompt is omitted if there are no substructures/superelements. Otherwise, the user specifies which superelement is to have nodes plotted. The superelements may be specified in any order and a superelement may be specified more than once, but only

the latest specifications for a given superelement are retained. The first superelement specification must not be negative, but on subsequent superelement specifications, a negative entry indicates that the same node plotting options as for the previous superelement are used and hence prompts (12)-(15) are omitted.

(12) THERE ARE III GEOMETRIC NODES.

This information is provided for the user's reference. In the case that there are substructures/superelements, this information applies to the specified superelements.

(13) ENTER FIRST & LAST NODES TO BE NUMBERED & INCREMENT
(ENTER 1,-1,1 TO NUMBER ALL NODES
ENTER 0, 0,0 IF NODE NUMBERING NOT DESIRED):

If there are substructures/superelements, this prompt applies to the substructure node numbers of the particular superelement.

(14) ENTER FIRST & LAST NODES TO BE PLOTTED & SYMBOL CODE
(ENTER 1,-1,1 TO PLOT ALL NODES WITH CIRCLES
ENTER 0, 0,0 IF NODE PLOTTING NOT DESIRED):

This prompt refers to the symbol to be plotted at each node coordinate.

(15) ENTER BOUNDARY NODE OPTION CODE & SYMBOL CODE
(OPTION <0 - BOUNDARY NODES - MASTER NODE NUMBERS
=0 - NO BOUNDARY NODES
>0 - BOUNDARY NODES - LOCAL NODE NUMBERS):

This prompt is omitted if there are no substructures/superelements. Otherwise, the user defines how the boundary nodes are

to be plotted and numbered. Note that only the sign (+ or - or 0) of the boundary node option code is important.

If there are substructures/superelements prompt (11) and prompts (12) to (15) (if applicable), are repeated until the user terminates node plotting specifications with a 0 entry at prompt (11). If there are no substructures/superelements, prompts (12) to (15) appear once only.

(16) DO YOU WANT A SUMMARY OF THE ELEMENTS AND GROUPS? (0=NO)

If the response is:

Element Group	Type of Elements	Number of Elements
xx	x	xxx

(16A) DO YOU WANT TO PLOT ANY ELEMENTS? (0=NO):

If the user wants any of the elements plotted, a non-zero response should be entered. If the user enters 0, prompts (17) to (19) are omitted.

(17) ENTER NUMBER OF SUPERELEMENT FOR ELEMENT PLOTTING
(<0 FOR SAME OPTIONS AS PREVIOUS SUPERELEMENT
=0 TO TERMINATE ELEMENT PLOTTING SPECIFICATIONS
>0 TO SPECIFY ELEMENT PLOTTING OPTIONS)

This prompt is omitted if there are no substructures/superelements. Otherwise, the user specifies which superelement is to have elements plotted. The superelements may be specified in any order and more than once but only the latest specification for a

given superelement is retained. The first superelement specification must not be negative, but on subsequent superelement specifications a negative entry indicates that the same element plotting specifications as for the previous superelement are to be used and hence prompts (18) to (19) are omitted.

(18) DO YOU WANT ALL OF THE ELEMENTS PLOTTED? (0=NO)

If the user wants all the elements to be plotted (in the specified superelement, if applicable), then a non-zero response causes prompt (19) to be omitted.

(19) SPECIFY ELEMENT GROUP AND FIRST AND LAST ELEMENTS
TO BE PLOTTED (ENTER X,1,-1, TO PLOT ALL ELEMENTS
OF GROUP X, AND 0,0,0 TO TERMINATE):

The user specifies which element group and elements (of the specified superelement, if applicable) are to be plotted. The element groups may be specified in any order, and element groups may be specified more than once, but only the latest specified range of elements is retained. If an element group is specified which does not exist, a warning is issued and the input is ignored.

Prompt (19) continues to reappear until the user enters an element plotting specification of 0,0,0. If there are substructures/superelements prompt (17) then reappears, and the cycle of prompts (17) to (19) repeats until the user enters a 0 superelement specification for prompt (17).

(20) * VIEW SPECIFICATIONS *

(21A) ENTER DIRECTION COSINES OF LINE OF VIEW,
ROTATION ANGLE, AND PLOT REDUCING FACTOR (PERCENT):

(21B) ENTER ROTATIONS ABOUT X, Y & Z AXES AND PLOT
REDUCING FACTOR (PERCENT):

The prompt which appears here depends on the response to (3). Prompt (21A) corresponds to the viewing vector approach and (21B) corresponds to the finite angular rotations.

The plot reducing factor defines the percentage reduction of the plot size. The plot reducing factor will be set to 30% if specified as less. This is necessary to provide space for the dialog area below the plot. The user is informed of this by the following message appearing:

"THE PLOT REDUCING FACTOR WILL BE SET TO 30 PERCENT."

(22A) ENTER ELEMENT TYPES TO BE PLOTTED
(=0 TO PLOT ALL TYPES
=1 TO SPECIFY PARTICULAR TYPES):

(22B) ENTER ELEMENT TYPE
(=0 TO TERMINATE):

This prompt is omitted if no element plotting is to be done. Otherwise, it allows the user to specify the type of elements to be plotted. Prompt (22A) appears first. If the user enters 0 in response to this prompt, all element types will be plotted and prompt (24B) is omitted. Otherwise, prompt (22A) appears, and the user enters the element type code (IEC) for the element types which are to be plotted. Only one type should be specified at a time, and prompt (22B) will appear after each entry. This cycle is terminated by entering 0.

(23) ENTER 0 FOR 2-DIMENSIONAL MODEL
1 FOR 3-DIMENSIONAL MODEL

This prompt is necessary to determine if repeated redisplay and redefinition of the window on different views of the model are required to specify node selection using the window capability. If '0' is entered then the selection of nodes using a window allows the user to specify boundary conditions for all the nodes located within the window. If '1' is entered the user is prompted to enter a second view. The nodes to be assigned boundary conditions are the nodes common to the recursive views.

(24) DO YOU WANT THE BOUNDARY CONDITIONS INDICATED? (0=NO):

This prompt appears if boundary condition data exists. If '0' is entered in response to Prompt 7 then this prompt does not appear for the first plot. It does appear after boundary condition data is generated and the plot specifications are redefined. The boundary conditions are displayed as an XYZ coordinate axis triad at the nodes at which they are applied. Translation and rotation are represented by crosses and circles, respectively at the middle of the appropriate triad axis. Note that for some elements rotation constraints are applied about the element local axes. For clarity, VASTBC indicates the constraints about the global axis. In effect then, VASTBC indicates a constraint of degree-of-freedom four as circle about the global X axis, degree-of-freedom five about the global Y axis and degree-of-freedom six about the global Z axis.

(25) DO YOU WANT THE B.C. NODES NUMBERED? (0=NO):

This prompt appears if the boundary conditions are to be indicated. If the user replies in the affirmative, the node

numbers are plotted beside the triad at each node at which boundary conditions are applied.

(26) PRESS RETURN TO PLOT

(27) *** CURSOR OPTIONS ***

The screen is first erased and the the legend is displayed on the right hand side of the screen indicating the alphanumeric keyboard entry to use for the cursor options. Next, the plot appears on the screen as specified by the responses to prompts (9) to (25). Plotting is finished when the borders and title appear. A single bell will sound to indicate that the program is expecting input. Any alphanumeric keyboard entry, followed by 'Return', or simply 'Return' will cause the screen to be cleared and control to be transferred to 29. Otherwise, the user may use the graphics cursors options described below.

GRAPHICS CURSOR OPTIONS: These options are selected by entering one of the following in response to the single bell:

I = select individual nodes
M = select multiple nodes
A = assign stiffness codes to selected nodes
S = specify skew coordinates for selected nodes
L = label nodes
W = window
R = recover original plot

Any other keyboard entry, or simply 'Return', causes control to transfer to (29). These cursor options are described in the following sections. The L, W and R options are the same feature available in VAST graphics program PLOTV1.

(28A) I = Select individual nodes with cursor.

This option allows the user to select individual nodes using the graphics cursor. If the user selects this option, a bell will sound, indicating that input is expected via the graphics cursor. The user then positions the cursor near the node of interest and enters any alphanumeric keyboard character or simply 'Return'. Once the node is selected a bell will sound and the cycle is repeated. The cycle is broken when the user selects a point outside the plot (i.e. a position in the legend). The selection of such a point causes control to return to (27). The user is notified of the change in control by the following message appearing in the dialog area.

ENTER CARRIAGE RETURN AND FOLLOW LEGEND ON THE RIGHT.

The user should note that the program searches through all the node coordinate data to find the node(s) closest to the cursor position. If there are substructures/superelements, all the substructures and associated substructure numbers are recorded for the selected locations. The stiffness codes for these nodes may be assigned by entering an 'A' when control returns to (27).

(28B) M = Select multiple nodes with cursor.

This option allows the user to select multiple nodes using a window. If the user selects this option a bell will sound indicating that input is expected via the graphics cursor. The user positions the cursor at the lower left hand corner of the area to be windowed, and enters any alphanumeric character, or simply 'return'. A bell will sound and the user repositions the cursor to the upper righthand corner of the area to be windowed. A box is drawn around the area and a bell sounds. The user then enters a carriage return.

If the user has entered '0' to prompt (23) defining the model as 2-D then control is transferred ahead to (28C). The user then assigns stiffness codes for the nodes windowed and control returns to (27) and the user may enter an M to define another window. The same plot remains.

If the user entered '1' to prompt (23) defining the mode as 3-D then the user is prompted for a new view and the model is replotted. This allows further refinement of the node point selection as obtained by repeated redisplay and redefinition of the window, presumably on different model views. The stiffness nodes for these nodes may be assigned by entering an 'A' when control returns to (27).

(28C) A = assign stiffness codes for nodes selected.

This option allows the user to assign stiffness codes for nodes selected either individually using the cursor or multiple using a window. The following prompts to specify this data appear in the dialog area. The user is first prompted whether the same stiffness codes are desired for all the nodes identified.

DO YOU WANT TO ASSIGN THE SAME DEGREE OF FREEDOM STIFFNESS CODES FOR ALL NODES IDENTIFIED? (0=NO, 1=YES)

The next prompt is for the user to enter the codes.

INPUT 6 CODES SPECIFYING WHICH DEGREES OF FREEDOM ARE TO BE ASSIGNED STIFFNESS.

If nodal points have been selected which already have stiffness codes assigned then the user is notified and asked if they are to be changed.

STIFFNESS CODES ALREADY SPECIFIED FOR NODE: XX
ENTER: 0=NO CHANGE, 1=NEW DATA, 2=REMOVE CONSTRAINT

The user is notified when all the nodes selected have been assigned by the following message in the display area:

ALL NODES SELECTED HAVE HAD CODES SPECIFIED INPUT A CARRIAGE RETURN. THEN FOLLOW LEGEND ON THE RIGHT.

Control returns to (27). If the model is substructured then the assigning of a boundary condition to a substructure node will result in node being made a master node if the node was not already specified as such. The superelement data is updated to reflect this change.

(28D) S = Skew Data.

This option allows the user to identify the node point requiring a skew coordinate and to input the skew data to define the skew. If the user selects this option a bell will sound, indicating input is expected for the cursor. The user then positions the cursor near the node of interest and enters any alphanumeric keyboard character or simply 'Return'. Once the node is selected then next action depends upon the response entered to Prompt (8).

If '0' was entered then the user is prompted via the dialog area to enter the rotation in degrees for the skew coordinate.

ENTER ROTATION IN DEGREES FOR SKEW TO BE ASSIGNED TO NODE:XX

If '1' was entered then a bell sounds indicating that input is expected via the graphics cursor. The user then positions the cursor at a nodal point perpendicular to the node selected. The

axis of the skew coordinate system is calculated to pass through these two points, and is assumed positive in the direction from the nodal point to the second point. The calculated rotation in degrees then appears in the dialog area.

CALCULATED ROTATION DEGREES : A TO BE ASSIGNED TO NODE:XX
ENTER A CARRIAGE RETURN TO CONTINUE WITH THE CURSOR AND TO STOP - SELECT A NODE POINT OUTSIDE THE PLOT.

The user is informed for both options that the cycle of specifying skews may be repeated until a nodal point is selected outside the plot.

ENTER CARRIAGE RETURN THEN FOLLOW LEGEND.

(28E) Node Labelling.

This option allows the user to selectively number nodes using the graphics cursor. If the user selects this option, two bells will sound, indicating that input is expected via the graphics cursor. The user then positions the cursor near the node of interest and enters any alphanumeric keyboard character or simply 'Return'. If a keyboard character is entered, do not follow it with 'Return'. The program then searches through all of the node coordinate data to find the node(s) closest to the cursor position. The node position is marked with a circle, and the node number appears at the cursor position. If there are substructures/superelements, both the superelement number and associated substructure node number will appear. Once the node is numbered, two bells will sound and the cycle is repeated. The cycle is broken when the user selects a point outside the screen window. The selection of such a point causes the control to return to (27). The user is notified of the change in control by the following message appearing in the dialog area.

ENTER A CARRIAGE RETURN AND THEN FOLLOW LEGEND ON THE RIGHT.

The user should note that since the computer uses the node coordinates and cursor coordinates as real numbers, it is unlikely that it will determine any two nodes as equal distance from the cursor, even if they should be. Thus, although VASTBC will number almost any number of nodes if the computer determines they are all an equal (minimum) distance from the cursor, it will generally be difficult for the user to get more than one node number for a single cursor position.

(28E) W = Windowing.

This option allows the user to window the plot. This may be used to blow up or expand a portion of the plot, or to reduce the plot. If the user selects this option, two bells will sound, indicating that input is expected via the graphics cursor. The user positions the cursor at the lower left hand corner of the area to be windowed, and enters any alphanumeric keyboard character, or simply 'Return'. If a keyboard character is entered, do <u>not</u> follow it with return. Two bells will sound and the procedure is repeated for the upper right hand corner of the area to be windowed. A box is then drawn around the area to be windowed, and a single bell will sound. When the user enters any keyboard character followed by 'Return', or simply 'Return', control is transferred back to (27), with only the windowed portion of the plot being displayed. Note that if the user attempts to reduce the plot by windowing outside it, the box around the windowed area will not appear, as it will lie outside the present virtual coordinate limits. This usage, however, is still valid. Also, if the user has numbered the nodes selectively before windowing, these node numbers will not appear on the windowed plot, but may be renumbered again, after the plot is finished, using the technique described in (28E).

(28F) R = Recovery of original plot.

If the user selects to recover the original plot, which is the first plot which appeared after the present plot and view parameters were specified, control is transferred back to (27) and the user may therefore select another cursor option as described in the legend after the plot is completed.

(29) ENTER 0 TO TERMINATE
1 TO CHANGE VIEW SPECIFICATIONS
2 TO CHANGE PLOT SPECIFICATIONS
3 TO CHANGE ALL SPECIFICATIONS:

An entry of 1 allows only the view specifications (prompts (20) to (25)) to be changed, an entry of 2 allows only the plot specifications (prompts (9) to (19)) to be changed, and an entry of 3 allows both to be changed.

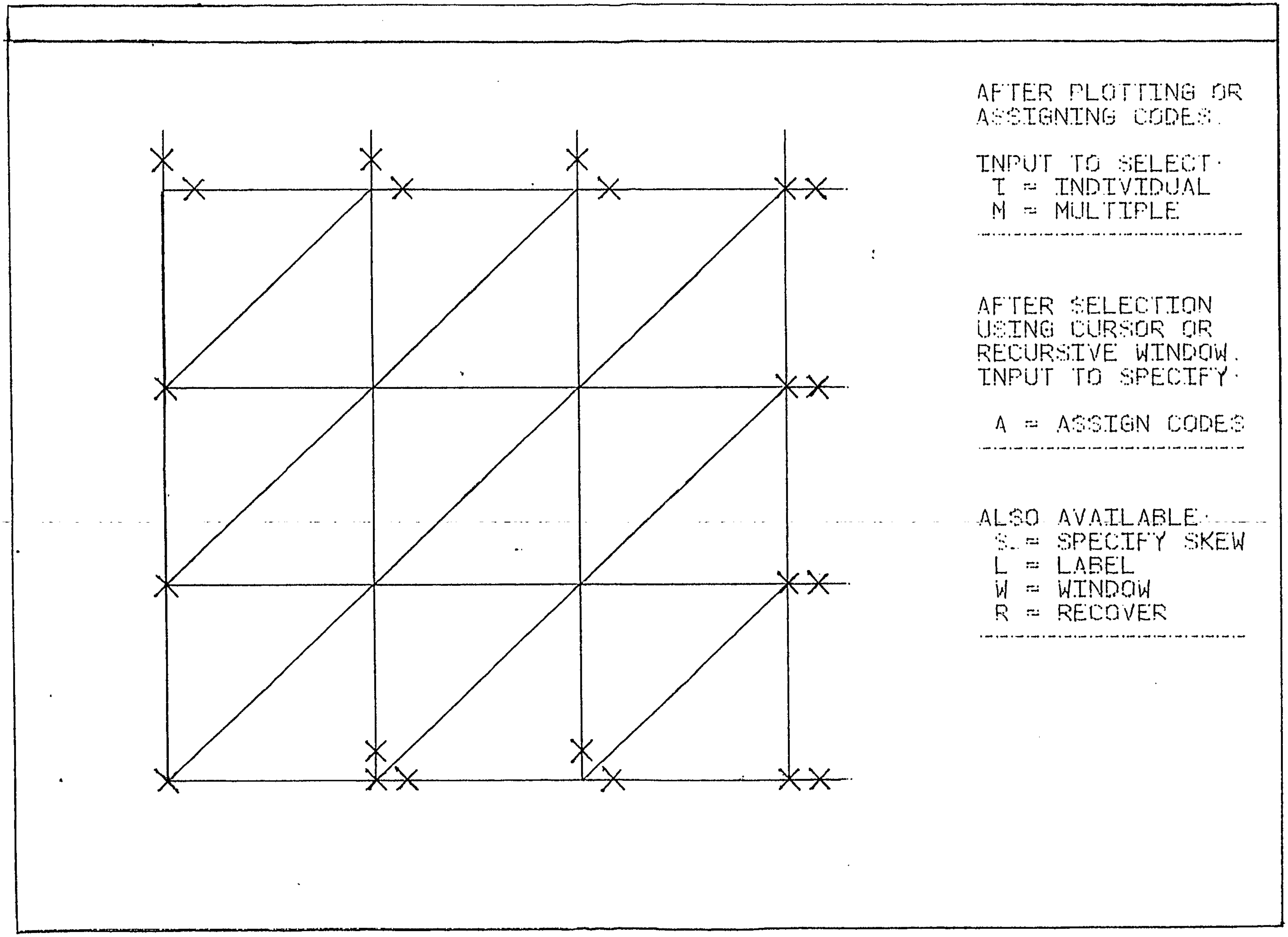


FIGURE A.1: Current Options Available in VASTBC

APPENDIX B

PROGRAM HULLD1

HULLD1

HULLD1 generates a VAST load file for edge pressures on a panel composed of VAST element types 1 and 4. The user is provided with the option of specifying normal edge pressures and/or shear pressures. The loading conditions may be specified at each nodal point, or as a linear distribution along an edge or as a quadratic distribution along an edge. The load for normal loading results in element surface load data being generated and nodal concentrated force data being generated for shear loading.

The program requires as input VAST geometry file (PREFX.USE and/or PREFX.GOM and/or PREFX.SED). The program generates a VAST PREFX.LOD file.

Operating Instructions for Program HULLD1

Files Required: PREFX.USE, PREFX.GOM, PREFX.SED

Terminal Response:
(Note: all input is in free format unless otherwise specified.)

(1) PROGRAM HULLD1: GENERATES VAST05 LOAD FILE FOR EDGE PRESSURES ON A PANEL (ELEMENT SURFACE LOADING DATA FROM NORMAL PRESSURE LOADING AND NODAL CONCENTRATED FORCE DATA FROM SHEAR LOADING)

(2) ENTER THE VAST FILE NAME PREFIX

The user inputs the VAST filename 'PREFX'. This is the prefix name which identifies the data file sets, and may be up to 5 characters long. HULLD1 searches for the data files with the specified program name.

(3) DO YOU WANT TO SELECT THE NODES FOR LOADING GRAPHICALLY? (0=NO, 1=YES)

The option to select nodes graphically is currently not available in HULLD1.

(4) ENTER 1 TO SPECIFY NORMAL PRESSURES ONLY
2 TO SPECIFY SHEAR PRESSURES ONLY
3 TO SPECIFY NORMAL AND SHEAR PRESSURES

This allows the user to specify the type of loading data that is to be generated by the HULLD1 program. The program will generate loading for VAST element types 1 and 4 only.

NOTE: Statements (1) to (4) appear only once, at the beginning of each execution of HULLD1. The parameters specified in response to these prompts may only be changed by restarting HULLD1.

(5) SUPERELEMENT:XXX
ENTER 1 TO LOAD THIS SUPERELEMENT
0 NOT TO LOAD SUPERELEMENT

This prompt is omitted if there are no substructures/super-elements. Otherwise the user specifies if the superelement is to be loaded.

(6) ***SELECT NODES AND SPECIFY LOADING***

(7) ENTER 1 TO SPECIFY THE NODE NUMBERS INDIVIDUALLY
2 TO SPECIFY THE NODE NUMBERS USING NODE GENERATION
0 TO TERMINATE

The user is provided with the above options to identify nodal points which are to be loaded. If the user enters 0 in response to this prompt then control goes forward to (13).

(8A) ENTER THE NUMBER OF NODES FOLLOWED BY THE NODE NUMBERS. NODES MUST BE INPUT SEQUENTIALLY ALONG AN EDGE, OTHERWISE PRESSURES MUST BE SPECIFIED FOR EACH POINT INDIVIDUALLY.

(8B) ENTER THE FIRST AND LAST NODE NUMBER AND THE INCREMENT BETWEEN EACH NODE.

The prompt which appears depends upon the response to (7). Prompt (8A) corresponds to specifying the nodes individually and (8B) corresponds to specifying the nodes using node generation.

(9) ENTER 1 TO DEFINE THE SHEAR PRESSURE TO BE IN THE X-DIRECTION
2 TO DEFINE THE SHEAR PRESSURE TO BE IN THE Y-DIRECTION

If the response to (4) indicated that shear pressure loading would be generated then this prompt will appear.

(10) ENTER 1 TO SPECIFY THE PRESSURE AT EACH NODE INDIVIDUALLY
2 TO SPECIFY THE PRESSURE AT THE END POINTS AND TO APPLY A LINEAR PRESSURE DISTRIBUTION
3 TO SPECIFY THE PRESSURE AT THE END POINTS AND A THIRD POINT (QUADRATIC PRESSURE DISTRIBUTION)

The user is provided with the above options to generate the load at nodal points specified.

(11A) ENTER NORMAL PRESSURE COMPONENT FOR NODE:XXX
ENTER THE SHEAR PRESSURE COMPONENT FOR NODE:XXX

If the response to (10) was 1, then the above prompts will appear depending upon the response to (4). These prompts will be repeatedly displayed until all the nodes identified are loaded and then control returns to (6).

(11B) ENTER THE NORMAL PRESSURE COMPONENT FOR THE FIRST NODE AND THE NORMAL COMPONENT FOR THE LAST NODE
ENTER THE SHEAR PRESSURE COMPONENT FOR THE FIRST NODE AND THE SHEAR PRESSURE COMPONENT FOR THE LAST NODE

If the response to (10) was 2, then the above prompts will appear depending upon the response to (4). After the loading for all the nodal points is calculated by the program then control returns to (6).

(11C) ENTER THE INTERMEDIATE NODE NUMBER:XXX
ENTER THE NORMAL PRESSURES FOR THE FIRST, MIDDLE, AND LAST NODES RESPECTIVELY
ENTER THE SHEAR PRESSURES FOR THE FIRST, MIDDLE, AND LAST NODES RESPECTIVELY

If the response to (10) was 3, then the above prompts will appear depending upon the response to (4). After the loading for all nodal points is calculated then control returns to (6).

(12A) In responding to prompts (11A) to (11C) if a node is found to be already loaded then the following prompt appears:

NODE:XXX IS ALREADY LOADED.
IS THIS A CORNER NODE FOR TWO LOADED EDGES? (0=NO, 1=YES)

(12B) DO YOU WANT TO SPECIFY A NEW LOAD CONDITION? (0=NO, 1=YES)
If the response to (12A) is 0 then the above prompt will appear.

(13) ***GENERATE NEW LOAD FILE***

If there are no substructures/superelements then the program terminates.

If there are substructures/superelements and the superelement being loaded is not the last then control returns to (5).

APPENDIX C

HVAST TEXT FILE

TEXT FILE FOR PROGRAM HVAST

C.1

20

A typical finite element analysis using HVAST consists of 3 phases:

PHASE I- Generation of finite element data using special-purpose procedures:
1. Geometry and element generation;
2. Load generation;
3. Boundary condition definition;
4. Hydrodynamic added mass formulation.

PHASE II- Execution of external general-purpose programs:
1. Graphical presentation of input data with VASTG/PATRAN;
2. Creation of PREFX.USE file with VASUSE;
3. Linear analysis with VAST;
4. Non-linear analysis with ADINA;
5. Fluid-structure interaction with USA;
6. Graphical presentation of results with VASTG/PATRAN.

PHASE III- Post-processing of results using special-purpose procedures:
1. Displacement plotting;
2. Stress plotting.

4

HVAST offers the following MODEL TYPES for the analysis of ship structures:
(NOTE: An * indicates those presently available on the VAX)

```
* 1-   EQUIVALENT BEAM MODEL OF HULL GIRDER
  2-   EQUIVALENT 2-D MODEL OF HULL GIRDER
* 3-   3-D MODEL OF HULL GIRDER
  4-   2-D/3-D MODELS OF COMPONENT STRUCTURES
  5-   2-D MODELS OF DECKS/BULKHEADS/ETC.
  6-   STIFFENED PANEL MODELS
  7-   RUDDER/STABILIZER
  8-   PROPELLER SHAFT MODEL
* 9-   LATTICE MAST
```

7

HVAST is organized to conduct a finite element analysis in a step-wise procedure. In general, the ANALYSIS STEPS must be carried out sequentially. When this is not the case, the user will be informed. The ANALYSIS STEPS are listed below. Note: Some of the steps do not apply to all MODEL TYPES.
(NOTE: An * indicates those presently available on the VAX)

```
* 1-   DISCRETIZATION OF STRUCTURE INTO FINITE ELEMENT MODEL

  2-   SPECIFICATION OF SKEWED COORDINATES/MULTI-POINT CONSTRAINTS

  3-   DEFINITION OF BOUNDARY CONDITIONS

  4-   ADDITION OF LUMPED MASSES

* 5-   HYDRODYNAMIC ADDED MASS MODELLING

  6-   PRIMARY BENDING LOADS

* 7-   HYDROSTATIC/HYDRODYNAMIC PRESSURE LOADING

* 8-   WIND/AIR BLAST/UNDERWATER EXPLOSION LOADING

  9-   OTHER TYPES OF LOADING
```

1

-Loads from edge stress distributions

10- DIGITIZED DYNAMIC LOADING

* 11- INERTIAL/CONCENTRATED/SUPPORT MOTION LOADING

12- SPECIAL MODELLING OPTIONS
4

-ASSEMBLY OF INDIVIDUAL MODELS AS SUBSTRUCTURES
-TOP-DOWN MODELLING
-BOTTOM-UP MODELLING
-MESH REFINEMENT

13- PLOT OF FINITE ELEMENT MODELS

* 14- PLOT OF APPLIED LOADS

15- OTHER TYPES OF PLOTS
2

-Longitudinal weight distribution
-Load distribution as specified transverse sections of hull

* 16- EXECUTION OF EXTERNAL PROGRAMS
4

-VASTG/PATRAN (Plotting)
-VAST (Linear solution)
-ADINA (Nonlinear solution)
-USA (Fluid-structure interaction)

* 17- DISPLACEMENTS PLOTTING

* 18- STRESSES PLOTTING

15
Depending on the MODEL TYPE, the structural geometry and material properties are supplied to HVAST in one of the following three ways:

1. Detailed DATA BASE files created by the external programs DIGFEM and HVDBG2. More information on the content and format of these files is given in the HVAST Reference Manual.

2. Interactively and/or by the formatted data file PREFX.HVCR (described in the HVAST Reference Manual). The PREFX.HVCR can contain a number of data sets, each set being preceeded by its own unique header.

3. By the PREFX.GOM/SED file(s) described in the VAST User's Manual, created either manually, or interactively using model generating programs such as VASGEN, SVAST, PATRAN/PATVAS, etc.

6
The correlation between analysis steps and model types is provided by the following APPLICATION MATRIX.

MODEL TYPE	ANALYSIS STEP 1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18
1	1	0	0	0	1	0	0	1	0	-1	0	0	-1	1	-1	1	1	3
2	-1	0	0	0	1	0	0	1	0	1	0	0	-1	1	-1	1	0	0
3	-1	-1	1	-1	-1	0	4	1	0	0	-1	-2	-1	1	-3	1	-1	-5
4	-1	1	1	1	0	0	1	0	1	0	1	2	1	1	0	1	0	1
5	-1	1	1	1	0	0	0	0	1	1	1	0	1	1	0	1	0	1
6	-1	0	1	1	0	0	0	1	1	1	1	0	1	1	0	1	0	1
7	-1	0	1	1	1	0	1	0	0	1	1	0	1	1	0	1	0	0
8	-1	0	1	1	0	0	0	0	0	0	1	0	1	1	0	1	0	1
9	1	0	1	1	0	0	0	2	0	1	1	0	1	1	0	1	1	0

2
NOTES:

1 DISCRETIZATION OF STRUCTURE INTO FINITE ELEMENT MODEL

21 EQUIVALENT BEAM MODEL OF HULL GIRDER

This feature of HVAST can be used to create an equivalent beam model of all or part of a hull girder. The file PREFX.GOM is created. Also, the options are available to create: (i) a boundary condition file PREFX.SMD, which includes, for surface ships, stiffness terms accounting for the loss of buoyancy due to upward displacement -thus including the rigid body heave and pitch modes; and (ii), a mass file PREFX.MMD, containing structural and/or non-structural masses.

In order to create the file PREFX.GOM, structural geometry must be supplied either by file PREFX.HVCR (header EBGEOM), or by file EB-----.GOM (generated by the program SCRAP). The creation of a boundary condition file including stiffness terms accounting for buoyancy effects requires hull form data supplied either by file PREFX.HVCR (header EBFRM) or by file EB----.FRM (generated by SCRAP). Data for creating the mass file PREFX.MMD must be supplied either by file PREFX.HVCR (header EBMAS) or by file EB----.MAS (generated by SCRAP). For details on the PREFX.HVCR input files, see Section 1.1 of the HVAST Reference Manual. Information on the SCRAP generated files can be found in the manual for the SCRAP program

For more information, see Section 1.1 of the Reference Manual.

28 2-D MODEL OF HULL GIRDER

The 2-D model of a hull girder is generated using the DATA BASE files TG---.---, SH---.TMP, SH---.TTA AND SH---.TBS created by the programs DIGFEM and HVDGB2. (Note: DIGFEM and HVDBG2 are external to HVAST).

The procedure works on the theory that all components of the ship resisting axial loading will be modelled as bars (including all longitudinal beams and decks), and all components resisting shear are modelled as membrane elements. In the case of the sloping hull shell, it is assumed that only a portion of the thickness resists shear while the remainder resists axial loads.

Two transverse sections are used at a time. For each section, bar areas at vertical nodes and membrane thickness between the nodes are calculated and presented on a plot. The user then uses the graphics cursor to specify the connectivities for the bar and membrane elements based on the information shown on the plot. Elements may be added or deleted if desired. Once the elements have been generated between the two sections, a new section is introduced and the process repeated. The user is not required to connect all corresponding nodes between sections. This permits control over the fineness of the mesh. All nodes which are not included in any bar elements will have their areas added to the nearest node which is included in an element. These areas are adjusted so as to maintain the proper moment of inertia of the section.

If the user wishes to create the 2-D model in stages, a restart feature can be utilized.

For more information, see Section 1.2 of the HVAST Reference Manual.

20 3-D MODEL OF HULL GIRDER

The 3-D model of a hull girder is generated using the DATA BASE files TG----.---, TG----.MFL, SH----.TMP, SH----.TBS and SH----.TTA. Two methods are available for generating the 3-D model, depending on the source of the DATA BASE files:

Method I- Permits a detailed decsription of the longitudinal structure using the DATA BASE files created by pro-

gram DIGFEM.

Method II- Permits a simplified description of the longitudinal structure using the DATA BASE files created by program HVDBG2.

For more information,see Section 1.3 of the HVAST Reference Manual.

Note: -The DATA BASE generating programs DIGFEM and HVDBG2 are external to HVAST, and must be executed prior to execution of HVAST.
-At present, Method I must be carried out externally from HVAST, using the programs LSGEOM and SHIPVS.

0 2-D/3-D MODELS OF COMPONENT STRUCTURES
20 2-D MODELS OF DECKS/BULKHEADS/ETC.

This feature of HVAST provides for the generation of finite models of planar structures such as panels, bulkheads and decks. The detailed DATA BASE files are not required here. The data required may be input via the terminal keyboard, graphics tablet or the file PREFX.HVCR (header TDGEOM) (see Section 1.5 of the HVAST Reference Manual).

The models may consist of plate and beam elements, or membrane and bar elements. The grids are generated by first dividing the structure into patches which have unique shapes or characteritics. A grid is generated within each patch and then all the patches are assembled to give the final grid. The user may define straight sides by specifying 2 end nodes and curved sides by specifying 4 nodes along an edge.

A certain degree of planning is required by the user prior to running HVAST. This involves dividing the structure into suitable patches and deciding how the patch boundary nodes should be numbered and generated (the coordinates for some patch boundary nodes must be input directly while others may be generated automatically).

For more information, see Section 1.5 of the HVAST Reference Manual.

39 STIFFENED PANEL MODELS

This feature of HVAST provides for the generation of a finite element model of a flat stiffened panel. The input data may be supplied either interactively or by the file PREFX.HVCR (header SPGEOM) (see Section 1.6 of the HVAST Reference Manual).

The model of the stiffened panel may be composed 1 of 2 groups of element types. The first group consists of triangular plate bending and general beam elements, and the second group consists of thick/thin shell and isoparametric beam elements.

The panel laid out in the X-Y plane must be a quadrilateral, with straight sides but not necessarily rectangular in shape. Sides are referrred to as being either horizontal or vertical but because they may be skewed, it must be noted that the sides with the smallest angles with respect to the X axis is referred to as the horizontal sides.

The panel is laid out in grid form which makes it possible to vary the geometry (ie, panel thickness and beam cross-section properties) from grid point to grid point. The user may also specify constant properties for the panel.

Beam eccentricity normal to the panel may be specified. There are two methods of accounting for eccentricity: the use of special eccentricity options within the general/isoparametric beam elements; and the use of multi-point constraints (MPCs). When the MPC method

is used, new nodes are generated and their behaviour is made dependent on the behaviour of the original nodes. This method should be used only when necessary (eg, when ADINA is used as the solver). C.5

A capability to specify boundary conditions along the sides of the panel is included. A code must be entered for each side stating whether it will be free, simply supported, or clamped. Also, boundary conditions for taking advantage of symmetry can be generated. (Note: the option to interactively define boundary conditions (via Analysis Step #3) can still be utilized, if boundary conditions other than those described above are required.

For more information, see Section 6.1 of the HVAST Reference Manual.

35 RUDDER/STABILIZER MODELS

This feature of HVAST provides for the generation of a finite element model of a rudder or stabilizer , or similarly constructed structure. All structural and material properity data is input interactively by the terminal keyboard.

The structure is assumed to consist of 5 main components: spars; webs; plating; leading and trailing edge beams; and shaft. The webs and spars form a grid that is defined by the number of structural 'boxes' in both the chord-wise and span-wise directions. The Y coordinate axis lies along the connecting shaft axis, and the X and Z axes form the plane of symmetry. The webs must be parallel to the X-Z plane, but the spars need not be parallel to the Y-Z plane.

The geometry of the structure is defined by 3 sets of points. The upper and lower section profiles each consist of a number of points which define the X and Z coordinates of the section. Points at sections between these two are assumed to vary linearly between them. Point pairs must be specified at both lead and trailing edges (Z = 0.0). In addition, there must be a point pair wherever a spar is desired. Otherwise, any additional point pairs may be used to define the finite element grid in the chord-wise direction. (The number of point pairs must be the same for both the upper and lower section profiles. The third set of points consists of the Y coordinates of the finite element grid lines in the span-wise direction. The only restrriction on this set of points is that there must be a point for every web location.

The model uses membrane elements or shear web elements for the web and spars. Combined membrane/plate elements are used for the plating. The thickness of the webs or spars, or plating may vary from box to box. The section properties of the leading or trailing edge beams may vary between web boxes. The shaft is assumed to be circular, and must terminate at a web, rather than just a finite element grid line.

For more information, see Section 1.7 of the HVAST Reference Manual.

0 PROPELLER SHAFT MODEL
16 LATTICE MASTS

This feature of HVAST provides for the analysis of lattice masts of warships (or similiar structures). Structural details must be supplied to HVAST by 1 of 2 ways: either by a PREFX.GOM file prepared in accordance with the VAST User's Manual, or by the file PREFX.HVCR (header LMGEOM) as described in Appendix C (Cards 1-7) of the User's Manual for the engineering Standard for the Design of Lattice Masts of Ships - Vol II.

When supplying data via PREFX.HVCR, the mast is modelled using only general beam and triangular plate elements. If file PREFX.GOM is used, other types of elements can be used. However, all beam and plate

element groups should appear first in the data.

For more information, see Section 1.9 of the HVAST Reference Manual or the User's Manual for the Engineering Standard for the Design of Lattice Masts of Ships - Vol II.

2 SPECIFICATION OF SKEWED COORDINATES/MULTI-POINT CONSTRAINTS

0

2 DEFINITION OF BOUNDARY CONDITIONS

6

This feature of HVAST permits the user to define or to edit existing boundary condition data interactively using the graphics cursor. The user proceeds by specifying the nodes to be plotted,the elements to be plotted, viewing parameters and various other specifications. Boundary conditions may be indicated if they are contained on an old PREFX.SMD file, or plotted after they are generated.

2 ADDITION OF LUMPED MASSES

15

This feature permits the addition of lumped masses to the finite element model. The formatted data file PREFX.HVCR (see Section 3.1 of the HVAST Reference Manual) is required. The file PREFX.MMD is created.

The possibility that the coordinate axes systems of the lumped masses and the finite element model do not coincide is accounted for by using coordinate tranformation data provided by the user.

Two types of lumped mass are allowed-concentrated and distributed In the case of a concentrated mass, the program will apply it to the node closest to the point of application. In the case of a distributed mass, a circular patch enclosing the mass is used to distribute the mass to nodes enclosed by the patch.

For more information, see Section 4.1 of the HVAST Reference Manual.

1 HYDRODYNAMIC ADDED MASS

22 EQUIVALENT BEAM MODEL OF HULL GIRDER

This feature of HVAST permits the user to generate a hydrodynamic added mass matrix for an equivalent beam model of a hull girder. An added mass file PREFX.T36 is created.

Either a lumped or a consistent mass matrix can be generated. Ideally, a different lumped mass matrix should be generated for each natural mode calculated. This is a costly procedure and experience has shown that the matrix generated for the first bending mode will give very good results for the first several modes. The consistent added mass matrix has off-diagonal coupling terms, and therefore can be used for all modes. Generally, if a large number of modes are required, and/or the length to depth ratio of the hull is less than ten, a consistent mass should be used. If the lumped mass approach is to be used, then an appropiate matrix can be generated through the use of a "bending node number". For free-free vibration, the first natural mode refers to the 2 node bending mode.

The PREFX.GOM file is required. Also, the hull form data as supplied on file PREFX.HVCR (header EBFRM) or the file EB----.FRM (created by SCRAP).

For more information, see Section 5.1 of the HVAST Reference Manual.

19 2-D MODEL OF HULL GIRDER

This feature of HVAST permits the user to generate a hydrodynamic added mass matrix for a 2-D finite element model of a hull girder. An added mass file PREFX.T36 is created.

A lumped mass matrix is generated. Ideally, a different lumped mass matrix should be generated for each natural mode calculated. This is a costly procedure and experience has shown that the matrix generated for the first bending mode will give very good results for the first several modes when the length to depth ratio of the hull is ten or greater. If a fairly high order mode is required, and/or the aspect ratio is less than ten, the appropiate mass matrix can be generated through the use of a "bending node number". For free-free vibration, the first natural mode refers to the 2 node bending mode.

The PREFX.GOM file is required. Also, the hull form data as supplied on file PREFX.HVCR (header EBFRM) or the file EB----.FRM (created by SCRAP) must be available.

For more information, see Section 5.2 of the HVAST Reference Manual.

0 3-D MODEL OF HULL GIRDER
0 2-D/3-D MODELS OF COMPONENT STRUCTURES
0 2-D MODELS OF DECKS/BULKHEADS/ETC.
0 STIFFENED PANEL MODELS
15 RUDDER/STABLIZER MODELS

This feature of HVAST permits the user to generate a finite element model of the fluid surrounding a rudder/stablizer for the calculation of hydrodynamic added mass.

The finite element grid of the fluid in the X-Y plane is defined by the structural finite element grid. The fluid element grid is extended by 2 elements on all 4 perimeters, unless the user specifies that the structure is only partially submerged-in which case the fluid element grid is extended by 2 elements only at the leading and trailing edges and at the lowest web edge. The user may specify the number of layers of elements on either side of the structure (ie, in the Z direction). The same number of layers will be on either side, with a maximum of 5 layers per side, for a total of ten layers.

For more information, see Section 5.7 of the HVAST Reference Manual.

0 PROPELLER SHAFT MODEL
0 LATTICE MAST

1 PRIMARY BENDING LOADS
----
0 EQUIVALENT BEAM MODEL OF HULL GIRDER
0 2-D MODEL OF HULL GIRDER
0 3-D MODEL OF HULL GIRDER
0 2-D/3-D MODELS OF COMPONENT STRUCTURES
0 2-D MODELS OF DECKS/BULKHEADS/ETC.
0 STIFFENED PANELS
0 RUDDER/STABLIZER MODEL
0 PROPELLER SHAFT MODEL
0 LATTICE MAST

1 HYDROSTATIC/HYDRODYNAMIC PRESSURE LOADING
----
0 EQUIVALENT BEAM MODEL OF HULL GIRDER
0 2-D MODEL OF HULL GIRDER
54 3-D MODEL OF HULL GIRDER

This feature of HVAST permits the user to specify various types of loading for a 3-D model of a hull girder. The file PREFX.HVCR (with appropiate header) is used to supply the necessary data. The files PREFX.GOM and PREFX.SED (for SUBSTRUCTURE/SUPERELEMENT) are required also. This feature of HVAST is described in Section 7.3 of the HVAST Reference Manual. C.8

Four different types of loading are available. Type 1 is hydrostatic pressure. Using a water line specified by the user, pressures normal to the hull shell in the form of nodal point forces are generated. The user can request standard hydrodynamic pressure or DMEM10 hydrodynamic pressure. Data for load type 1 must be preceeded by the header H3DLD1.

Type 2 consists of hydrodynamic pressures applied to the hull shell in the form of pressure distributions applied normal to the hull cross sections at various longitudinal stations. Either static or dynamic loading can be specified. The user must describe a minimum of three pressure distributions along the length of the hull. A typical pressure distribution is described by a pressure vs angle curve, where the angle is defined relative to the intersection of the vertical Y axis and a horizontal line parallel to the X axis and passing through the water line. Nodal point forces are generated by interpolating the pressure curves in both the longitudinal direction and around the hull shell surface. Data for load type 2 must be preceeded by the header H3DLD2.

Load type 3 is similiar to load type 2, except that only a single pressure distribution is specified by the user, and the pressure at longitudinal stations are determined using multiplying factors which vary in the longitudinal direction. These factors must be supplied by the user for at least two longitudinal positions. Data for load type 3 must be preceeded by the header H3DLD3.

Load type 4 provides a method for the approximate determination of hull response due to slamming by assuming the pressure load is applied as a time varying force at the keel. Only a dynamic load can be generated. The user defines a time-dependent load function, and a multiplying factor curve which varies longitudinally. Loads at nodes located at the keel are generated by interpolation. Data for load type 4 must be preceeded by the header H3DLD4.

A PREFX.LOD file is created. Also, an auxiliary file named PREFX.HV34 for load type 1, 2 and 3 is created for plotting the pressure loads generated (Analyis Step 15).

The user indicates the desired load type as follows:

Option 1: Hydrostatic pressure;
Option 2: Hydrodynamic pressure (pressure distributions supplied at a number of sections);
Option 3: Hydrodynamic pressure (single pressure distribution and longitudinal multiplication factor supplied);
Option 4: Slamming approximation.

For more information, see Section 7.3 of the HVAST Reference Manual.

0 2-D/3-D MODELS OF COMPONENT STRUCTURES
0 2-D MODELS OF DECKS/BULKHEADS/ETC.
0 STIFFENED PANEL MODELS
11 RUDDER/STABILIZER MODELS

This feature of HVAST permits the user to specify pressure loads for the rudder/stablizer model. The file PREFX.HVCR (header RSLOD) (see Section 7.7 of the HVAST Reference Manual) is required.

Hydrostatic and/or user supplied pressure loading can be specified. The user supplied pressure can be either uniform (over the entire sur-

face) or non-uniform (permitting span-wise and chord-wise variation). In addition, the user supplied pressure can be presented in a time history form for a dynamic analysis.

For more information, see Section 7.7 of the HVAST Reference Manual.

0 PROPELLER SHAFT MODEL
0 LATTICE MASTS

1 WIND/AIR BLAST/UNDERWATER EXPLOSION LOADING
----
15 EQUIVALENT BEAM MODEL OF HULL GIRDER

This feature of HVAST provides for the generation of underwater explosion induced loading for the whipping analysis of a hull girder modelled as an equivalent beam. The file PREFX.LOD is created. Also, a PREFX.USE file is generated.

The calculation of bodily velocity time histories and plots of the velocity time histories and/or plots of the corresponding pseudo-velocoty shock spectrum can be obtained.

The PREFX.GOM file is required. Also, the hull form data as supplied on file PREFX.HVCR (header EBFRM) or the file EB----.FRM (created by SCRAP), and the hydrodynamic added mass matrix on file PREFX.T36 must be available.

For more information, see Section 8.1 of the HVAST Reference Manual.

0 2-D MODEL OF HULL GIRDER
0 3-D MODEL OF HULL GIRDER
0 2-D/3-D MODELS OF COMPONENT STRUCTURES
0 2-D MODELS OF DECKS/BULKHEADS/ETC.
0 STIFFENED PANEL MODELS
0 RUDDER/STABLIZER MODEL
0 PROPELLER SHAFT MODEL
37 LATTICE MAST

This feature of HVAST provides for the generation of wind or air blast loading on a lattice mast. Input data must be supplied by the file PREFX.HVCR (2 subsets, headers LMLOD1 and LMLOD2). This data is described in the User's Manual for the Engineering Standard for the Design of Lattice Masts of Ships - Vol II, subset LMLOD1 in Appendix D (Cards 1 and 2), and subset LMLOD2 in Appendix B (Cards 2-17).

In general, this step will require 2 separate runs. During the first run, pressure time-histories for various cylinder diameters and plate widths will be computed, using data supplied on subset LMLOD1 , including such air blast parameters as peak overpressuer, duration, ambient pressure and temperature, and decay constant. The results from this run can then be used to prepare (manually) the data subset LMLOD2.

During the second run, loads on the mast will be computed, using the data subset LMLOD2. The procedure for describing the air blast loading is rather general. Both dynamic pressures and diffraction phase pressure must be specified at various points in time. Constant or time varying drag coefficients can be specified. Wind loading can by accounted for, simply by providing a dynamic pressure for the desired wind velocity (diffraction is not a factor in wind loading). Generally, a static analysis is sufficient for wind loading. This can be accomplished by setting the number of time points equal to 1. The user has the option to have each member of the mast checked for vortex shedding. The option to bypass the calculation of air blast/ wind loading and do only the vortex shedding check is available.

The user identifies the run as follows:

Option 1: Computation of pressure time-histories for various cylinder diameters and plate widths;
OPtion 2: Generation of loads on mast.

For more information, see Section 8.9 of the HVAST Reference Manual or the User's Manual for the Engineering Standard for the Design of Lattice Masts of Ships - Vol II.

2 OTHER TYPES OF LOADING

0

2 DIGITIZED DYNAMIC LOADING

0

2 INERTIAL/CONCENTRATED/SUPPORT MOTION LOADS

10

This feature of HVAST provides a procedure for the user to input by terminal, the data necessary to create a PREFX.LOD file for inertial, concentrated and support motion loading.

If for inertial and/or concentrated loading, a PREFX.LOD file already exists, the user has the option to update it. Such is not the case for support motion, where a completely different PREFX.LOD file is needed, since this type of loading cannot be combined with other types.

For more information, see Section 11.0 of the HVAST Reference Manual.

2 COMBINING INDIVIDUAL MODELS FOR SUBSTRUCTURE ANALYSIS

0

2 PLOT OF FINITE ELEMENT MODELS

0

2 PLOT OF APPLIED LOADS

7

This feature of HVAST provides a procedure for plotting dynamic load time histories for user specified nodes. The nodes are idenified interactively.

The file PREFX.LOD is required as input.

For more information, see Section 14.1 of the HVAST Reference Manual.

1 OTHER TYPES OF PLOTS

11 EQUIVALENT BEAM MODEL OF HULL GIRDER

This feature of HVAST provides a procedure for plotting the longitudinal weight distribution of a hull girder modelled as an equivalent beam.

The user divides the model into a number of sections and the weight of each section is calculated. These section weights are then plotted against the length of the hull.

The files PREFX.GOM and PREFX.MMD are required as input.

For more information, see Section 15.1 of the HVAST Reference Manual.

10 2-D MODEL OF HULL GIRDER

This feature of HVAST provides a procedure for plotting the longitudinal weight distribution of a hull girder modelled as a 2-D model.

The user divides the model into a number of sections and the weight of each section is calculated. These section weights are then plotted against the length of the hull.

The files PREFX.GOM and PREFX.MMD are required as input.

For more information, see Section 15.2 of the HVAST Reference Manual.

11 3-D MODEL OF HULL GIRDER

This feature of HVAST provides a procedure for plotting the longitudinal weight distribution of a hull girder modelled as a 3-D model.

The user divides the model into a number of sections and the weight of each section is calculated. These section weights are then plotted against the length of the hull.

The files PREFX.GOM, PREFX.MMD and PREFX.SED (in the case of substructure/superelement analysis) are required as input.

For more information, see Section 15.3 of the HVAST Reference Manual.

0 2-D/3-D MODELS OF COMPONENT STRUCTURES
0 2-D MODELS OF DECKS/BULKHEADS/ETC.
0 STIFFENED PANEL MODELS
0 RUDDER/STABILIZER MODEL
0 PROPELLER SHAFT MODEL
0 LATTICE MAST

2 EXECUTION OF EXTERNAL PROGRAMS

51

This step in an HVAST analysis involves the execution of external programs. The programs to be executed depends on the type of analysis to be performed, and the extent to which the general purpose graphics programs are to be used for pre- and post-processing. The execution of external programs is divided into cateories A, B, C and D. These cateories are as follows:

A. Graphical presentation of input data:

There are two graphics packages available for plotting input data, VASTG and PATRAN. If the latter is used, the program VASPAT must be run first to translate the data into the PATRAN format. User's manuals for VASTG, VASPAT and PATRAN exist, and should be consulted if the user is not familiar with these programs.

B. Creation of PREFX.USE file.

This file contains all the data for a VAST or ADINA analysis which dosen't exist on the PREFX.GOM/SED/SMD/MMD/LOD files. It can be created manually, by consulting the VAST User's Manual, or it can be created interactively via the terminal by running the program VASUSE. It is recommended that VASUSE be used at all times.

C. Solution phase:

This involves executing the finite element solver program(s) to determine displacements, stresses, natural frequencies and/or buckling loads. The solver to be used depends on whether a linear or non-linear analysis is to be performed, and whether fluid-structure interaction is to be considered. If a linear analysis is to be performed, VAST should be used as the solver. If a non-linear analysis is to be performed, ADINA should be used. If ADINA is used, the program ADIDAT must be run first to translate the data into the ADINA format. For the case of fluid-structure interaction, the program USA must be used to compute nodal point displacements. User's manuals for VAST, ADINA, ADIDAT and USA exist, and should be consulted if the user in not familiar with these programs.

D. Graphical presentation of results:

There are two graphics packages available for plotting results, VASTG and PATRAN. If a linear analysis has been performed with VAST, the program VASPAT must be run before PATRAN can be used. If a non-linear analysis has been performed with ADINA, either the programs ADIPOS and VASPAT, or the program ADIPAT must be run prior to PATRAN. User's manuals for VASTG, PATRAN, ADIPOS, VASPAT and ADIPAT exist, and should be consulted if the user is not familiar with these programs.

1 DISPLACEMENT PLOTTING
----

7 EQUIVALENT BEAM MODEL OF HULL GIRDER

This feature of HVAST provides an interactive procedure for plotting node point displacement, velocity and acceleration time histories from an equivalent beam analysis.

The files PREFX.LOD and PREFX.T52 are required as input.

For more information, see Section 17.1 of the HVAST Reference Manual.

0 2-D MODEL OF HULL GIRDER
12 3-D MODEL OF HULL GIRDER

This feature of HVAST provides an interactive procedure for plotting hull static or dynamic displacements versus distance from the forward perpendicular, at specified water lines, and, in the case of a dynamic load, at specified time step.

Plots can be obtained for a model with or without substructures/superelements. In the case of the latter, the user must specify the first and last superelements which make up the total hull shell. (It is assumed that the superelements which define the hull shell form a continuous sequence.)

For more information, see Section 17.3 of the HVAST Reference Manual.

0 2-D/3-D MODELS OF COMPONENT STRUCTURES
0 2-D MODELS OF DECKS/BULKHEADS/ETC.
0 STIFFENED PANEL MODELS
0 RUDDER/STABLIZER MODEL
0 PROPELLER SHAFT MODEL
7 LATTICE MAST

This feature of HVAST provides an interactive procedure for plotting node point displacement, velocity and acceleration time histories from a lattice mast analysis.

The files PREFX.LOD and PREFX.T52 are required as input.

For more information, see Section 17.1 of the HVAST Reference Manual.

1 STRESS PLOTTING

25 EQUIVALENT BEAM MODEL OF HULL GIRDER

This feature of HVAST provides procedures for plotting beam stresses and stress resultants (moments) obtained from a finite element analysis. At present, three types of plots are available.

Option number one provides a procedure for graghicaly displaying combined axial and bending stresses for an equivalent beam analysis of the hull girder. Locations on the section where stresses are to be computed and displayed are specified interactively via cursor cross-hairs. NOTE: this feature cannot be used unless the detailed DATA BASE files TG----.--- created by DIGFEM or HVDGB2 are available.

Option number two provides a procedure for plotting bending moment time histories obtained from a dynamic analysis.

Option number three provides a procedure for plotting displaying maximum bending stresses in beams. At present, this feature is limited to static loading only.

The user indicates the desired plot as follows:

Option 1: Combined axial and bending stresses;
Option 2: Bending moment time histories;
Option 3: Maximum bending stress.

For more information, see Section 18.1 of the HVAST Reference Manual.

0 2-D MODEL OF HULL GIRDER
0 3-D MODEL OF HULL GIRDER
0 2-D/3-D MODELS OF COMPONENT STRUCTURES
0 2-D MODELS OF DECKS/BULKHEADS/ETC.
0 STIFFENED PANEL MODELS
0 RUDDER/STABLIZER MODEL
0 PROPELLER SHAFT MODEL
0 LATTICE MAST

APPENDIX D

SAMPLE HVAST TERMINAL SESSION

```
$ RUN HVAST01

PROGRAM HVAST:      COMPUTER PROGRAM FOR THE FINITE ELEMENT
                    ANALYSIS OF SHIP STRUCTURES.

                    VERSION #01    (DATE: 11-05-1989)


   THIS PRGORAM IS DESIGNED TO ELIMINATE, AS MUCH AS POSSIBLE,
THE USE OF A MANUAL TO DESCRIBE THE VARIOUS FEATURES OF HVAST.
CONSEQUENTIALLY, A LOT OF DIALOG CAN APPEAR ON THE SCREEN.  TO
FACILITATE READING, A PAUSE FEATURE HAS BEEN INCORPORATED. THE
PAUSE IN INDICATED WHEN THE SYMBOL "//" APPEARS AT THE LEFT OF
THE SCREEN. TO CONTINUE, TYPE ANY CHARACTER AND RETURN.

//


THERE ARE FOUR MODES OF EXECUTION:

   1- COMPUTATION  MODE (WITH MINIMUM DIALOG)
   2- COMPUTATION  MODE (WITH LIMITED DIALOG)
   3- COMPUTATION  MODE (WITH FULL DIALOG)
   4- HVAST REVIEW MODE (WITH FULL DIALOG)

ENTER DESIRED MODE
1

WHAT IS THE LINE SPEED?
9600

IDENTIFY TERMINAL TYPE ACCORDING TO RESOLUTION,
CURSOR AND COLOUR CAPABILITY:
ENTER 0 FOR TEKTRONIX 4006 OR EQUAL (LOW RES/NO CURS/NO COL)
      1 FOR TEKTRONIX 401* OR EQUAL (LOW RES/CURSORS/NO COL)
      2 FOR TEKTRONIX 4014/15 (HI RES/CURSORS/NO COL)
      3 FOR TEKTRONIX 4113 (COLOUR):
2

IDENTIFY TERMINAL TYPE ACCORDING TO DIALOG CAPABILITY:
ENTER 0 NO DIALOG AREA
      1 DIALOG AREA

0

ENTER THE VAST 5-CHARACTER JOB NAME PREFIX:
EBMEN

Specify MODEL TYPE for this analysis:
(If 0 entered, will be listed)

0

      1-   EQUIVALENT BEAM MODEL OF HULL GIRDER
      2-   EQUIVALENT 2-D MODEL OF HULL GIRDER
      3-   3-D MODEL OF HULL GIRDER
      9-   LATTICE MAST
//
```

```
Specify MODEL TYPE for this analysis:
(If 0 entered, will be listed)

1

THE FOLLOWING MODEL TYPE HAS BEEN SELECTED:

     1-   EQUIVALENT BEAM MODEL OF HULL GIRDER
//

THE ANALYSIS STEPS WHICH CAN BE PERFORMED FOR THE MODEL TYPE
SELECTED ARE AS FOLLOWS:

     1-   DISCRETIZATION OF STRUCTURE INTO FINITE ELEMENT MODEL
     3-   DEFINITION OF BOUNDARY CONDITIONS
     5-   HYDRODYNAMIC ADDED MASS MODELLING
     8-   WIND/AIR BLAST/UNDERWATER EXPLOSION LOADING
     10-  DIGITIZED DYNAMIC LOADING
     14-  PLOT OF APPLIED LOADS
     15-  OTHER TYPES OF PLOTS
     16-  EXECUTION OF EXTERNAL PROGRAMS
     18-  STRESSES PLOTTING
//

SPECIFY STEP TO BE PERFORMED. (TERMINATE EXECUTION BY ENTERING A ZERO)
1

THE FOLLOWING ANALYSIS STEP HAS BEEN SELECTED:

     1-   DISCRETIZATION OF STRUCTURE INTO FINITE ELEMENT MODEL
//

ENTER 1 TO CONTINUE EXECUTION OF THIS STEP, OR 0 TO BYPASS

1

PROGRAM HVEB -   GENERATES PREFX.GOM, PREFX.SMD AND PREFX.MMD
                 FILES FOR AN EQUIVALENT BEAM MODEL OF A SHIP

DO YOU WISH TO CREATE THE EBMEN.GOM FILE? (1=YES; 0=NO)
1

INDICATE METHOD BY WHICH STRUCTURAL GEOMETRY IT TO BE SUPPLIED:
(1=EBMEN.HVCR FILE; 2=EB----.GOM FILE)
1

FOR UNITS:    IN    FT    MM    M
ENTER    :    1     2     3     4
( FOR 1 AND 2 FORCE UNITS ARE LB
  FOR 3 AND 4 FORCE UNITS ARE N)
  (RELATES TO UNITS OF FINITE ELEMENT MODEL)
4

*EBMEN.GOM  FILE HAS BEEN CREATED
```

```
DO YOU WISH TO CREATE THE EBMEN.SMD FILE? (1=YES; 0=NO)
1

DO YOU WISH TO INCLUDE STIFFNESS TERMS ACCOUNTING FOR BUOYANCY
EFFECTS? (1=YES; 0=NO)
1

INDICATE METHOD BY WHICH HULL FORM DATA IS TO BE SUPPLIED:
(1=EBMEN.HVCR; 2=EB----.FRM)
1

ENTER DENSITY OF FLUID MEDIUM(FORCE/LENGTH**3/G)
(FOR DEFAULT VALUE FOR SEAWATER, ENTER 0 FOR DENSITY)
0

IS MODEL FULLY SUBMERGED?(1-Y,0-N)
0

ENTER 1 TO PERMIT BOTH VERTICAL AND HORIZONTAL MOTION
      2 TO PERMIT VERTICAL MOTION ONLY
      3 TO PERMIT HORIZONTAL MOTION ONLY
2

*EBMEN.SMD  FILE HAS BEEN CREATED

DO YOU WISH TO CREATE THE EBMEN.MMD FILE? (1=YES; 0=NO)
1

INDICATE METHOD BY WHICH MASS DATA IS TO BE SUPPLIED:
(1=EBMEN.HVCR; 2=EB----.MAS)
1

*EBMEN.MMD  FILE HAS BEEN CREATED

END OF STEP    1

ENTER 1 TO PERFORM NEW STEP, OR 0 TO TERMINATE EXECUTION

1

THE ANALYSIS STEPS WHICH CAN BE PERFORMED FOR THE MODEL TYPE
SELECTED ARE AS FOLLOWS:

     1-   DISCRETIZATION OF STRUCTURE INTO FINITE ELEMENT MODEL
     3-   DEFINITION OF BOUNDARY CONDITIONS
     5-   HYDRODYNAMIC ADDED MASS MODELLING
     8-   WIND/AIR BLAST/UNDERWATER EXPLOSION LOADING
    10-   DIGITIZED DYNAMIC LOADING
    14-   PLOT OF APPLIED LOADS
    15-   OTHER TYPES OF PLOTS
    16-   EXECUTION OF EXTERNAL PROGRAMS
    18-   STRESSES PLOTTING
//

SPECIFY STEP TO BE PERFORMED. (TERMINATE EXECUTION BY ENTERING A ZERO)
5
```

```
THE FOLLOWING ANALYSIS STEP HAS BEEN SELECTED:

     5-   HYDRODYNAMIC ADDED MASS MODELLING
//

ENTER 1 TO CONTINUE EXECUTION OF THIS STEP, OR 0 TO BYPASS

1

PROGRAM HVAM2 -  GENERATES HYDRODYNAMIC ADDED MASS OF A HULL
                 GIRDER REPRESENTED AS AN EQUIVALENT BEAM OR
                 A 2-D MODEL

ENTER 1 TO GENERATE LUMPED MASS MATRIX
   OR 2 TO GENERATE CONSISTENT MASS MATRIX
1

ENTER DESIRED MODE NUMBER
2

*EBMEN.T36 HAS BEEN CREATED

END OF STEP   5

ENTER 1 TO PERFORM NEW STEP, OR 0 TO TERMINATE EXECUTION

1

THE ANALYSIS STEPS WHICH CAN BE PERFORMED FOR THE MODEL TYPE
SELECTED ARE AS FOLLOWS:

     1-   DISCRETIZATION OF STRUCTURE INTO FINITE ELEMENT MODEL
     3-   DEFINITION OF BOUNDARY CONDITIONS
     5-   HYDRODYNAMIC ADDED MASS MODELLING
     8-   WIND/AIR BLAST/UNDERWATER EXPLOSION LOADING
     10-  DIGITIZED DYNAMIC LOADING
     14-  PLOT OF APPLIED LOADS
     15-  OTHER TYPES OF PLOTS
     16-  EXECUTION OF EXTERNAL PROGRAMS
     18-  STRESSES PLOTTING
//

SPECIFY STEP TO BE PERFORMED. (TERMINATE EXECUTION BY ENTERING A ZERO)
8

THE FOLLOWING ANALYSIS STEP HAS BEEN SELECTED:

     8-   WIND/AIR BLAST/UNDERWATER EXPLOSION LOADING
//

ENTER 1 TO CONTINUE EXECUTION OF THIS STEP, OR 0 TO BYPASS

1
```

```
PROGRAM HVLD9 -   GENERATES A TIME VARYING LOAD ON AN EQUIV-
                  ALENT BEAM OR 2-D MODEL OF A HULL GIRDER
                  DUE TO A UNDERWATER EXPLOSION.

ENTER CHARGE WEIGHT IN NEWTONS
2000

ENTER CHARGE COORDINATES H, D AND L (IN METERS)
65 30 60

ENTER THE EXPLOSIVE TYPE:

          1 - TNT
          2 - HBX-1
          3 - PETN
          4 - COMP B (NOT AVAILABLE)
          5 - NUCLEAR

1

ENTER THE EQUATION CONSTANTS K1 TO K11 AND A1 TO A5

ARE DEFAULT VALUES FOR ALL THESE CONSTANTS TO BE USED?
(1-Y,0-N)
1

SHOCK WAVE PARAMETERS:
======================


STAND-OFF (TO CENTER OF SHIP) (METERS) ---------- 0.71589E+02
PEAK PRESSURE (KILO-PASCALS) -------------------- 0.27332E+04
DECAY CONSTANT (MILLI-SECONDS) ------------------ 0.83699E+00
ENERGY DENSITY (KILO-JOULES/SQUARE METER) ------- 0.10033E+03
IMPULSE DENSITY (NEWTON-SECOND/SQUARE METER) ---- 0.33172E+01
HULL SHOCK FACTOR ------------------------------- 0.71252E+00
KEEL SHOCK FACTOR ------------------------------- 0.43683E+00
KEEL SHOCK FACTOR AT BOW ------------------------ 0.30273E+00
KEEL SHOCK FACTOR AT STREN ---------------------- 0.30503E+00
PEAK BODILY VELOCITY (METERS/SECOND) ------------ 0.00000E+00
SURFACE CUT-OFF PERION (AT KEEL) (SECONDS) ------ 0.00000E+00


GAS BUBBLE PARAMETERS:
======================



DEEP BUBBLE APPROXIMATION:

MAXIMUM BUBBLE RADIUS=    0.196E+01 (METERS)
BUBBLE PERIOD        =    0.167E+01 (SECONDS)



 IS BUBBLE MIGRATION TO BE INCLUDED?
 (1=YES; 0=NO)
0
```

```
 ARE FREE SURFACE EFFECTS TO BE INCLUDED?
 (1=YES; 0=NO)
1

 ENTER DRAG COEFFICIENT CD (0 FOR DEFAULT)
0

 TIME HISTORY OF BUBBLE PARAMETERS TO BE
 PRINTED? (Y=1,N=0)
1

 NUMBER OF PULSES LIMITED TO            2 FOR
 THIS EXPLOSION

 CONTINUE? (Y=1,N=0)
1

 SPECIFY INTEGRATION CUTOFF TIME IN SECONDS:
 (ENTER 0 FOR DEFAULT VALUE)
0

 PULSE TRANSITION AT TIME STEP          114

EXPLOSION LOAD TIME HISTORY INFORMATION:

     TOTAL NUMBER OF POINTS IN TIME    =  169
     DURATION OF LOAD (SECONDS)        =  0.1657E+01

*EBMEN.LOD HAS BEEN CREATED

END OF STEP   8

ENTER 1 TO PERFORM NEW STEP, OR 0 TO TERMINATE EXECUTION

0

FORTRAN STOP
```

APPENDIX E

SAMPLE HVAST INPUT FILE

PREFX.HVCR Data File for Equivalent Beam Model

```
EBGEOM
 UNDERWATER SHOCK - EQUIVALENT BEAM
   43     1
 0.207E+12    0.3
    1    -4.8800      0.1260      0.1280      0.0919
    2    -4.2700      0.1680      0.2580      0.1600
    3    -3.6600      0.1950      0.4370      0.2380
    4    -3.0500      0.2330      0.7200      0.3420
    5    -2.4400      0.2720      1.0700      0.4740
    6    -1.8300      0.3010      1.4900      0.6030
    7    -1.2200      0.3700      2.5200      0.7700
    8    -0.6100      0.5280      6.3200      1.4500
    9     3.0500      0.6900     10.6000      2.5000
   10     4.8800      0.7100     11.0000      3.6000
   11     8.5300      0.6900     10.3000      4.7000
   12    10.4000      0.7740     11.5000      6.0700
   13    14.0000      0.8840     13.5000      8.9000
   14    21.3000      0.9350     14.4000     12.3000
   15    26.8000      0.9740     15.2000     14.4000
   16    28.7000      0.9870     15.7000     16.4000
   17    32.3000      0.9610     14.7000     19.3000
   18    39.3000      1.0100     15.0000     22.0000
   19    43.0000      1.0100     15.0000     22.0000
   20    44.8000      1.0100     15.0000     22.0000
   21    46.6000      1.2500     17.7000     29.6000
   22    48.5000      1.2500     17.7000     29.6000
   23    52.1000      1.2500     17.7000     29.6000
   24    57.6000      1.2300     19.2000     30.0000
   25    64.4000      1.1700     19.0000     28.9000
   26    68.0000      1.1900     17.4000     28.5000
   27    68.6000      1.1700     17.8000     28.1000
   28    69.8000      1.1800     19.5000     29.0000
   29    75.3000      1.2100     18.5000     29.8000
   30    78.9000      1.2000     18.7000     28.4000
   31    82.0000      1.1900     18.4000     27.7000
   32    88.1000      1.0300     12.5000     23.3000
   33    94.2000      0.8580      6.8200     18.1000
   34    97.2000      0.8450      6.3600     17.1000
   35   100.0000      0.5480      2.9900     10.1000
   36   103.0000      0.5480      2.9900     10.1000
   37   105.0000      0.8580      3.3400     12.1000
   38   107.0000      0.8580      3.3400     12.1000
   39   111.0000      0.8580      3.3400     12.1000
   40   112.0000      0.6970      2.5600      9.7800
   41   117.0000      0.5450      1.7100      7.5700
   42   121.0000      0.5160      1.5000      6.7800
   43   124.0000      0.5160      1.5000      6.7800
```

PREFX.HVCR (cont.)

```
HULFRM
 20  20.0
      0.0       6.49     0.113     0.223   0.50980
      6.49     12.98     0.216     0.223   0.58076
     12.98     19.47     0.326     0.236   0.63945
     19.47     25.96     0.436     0.236   0.67822
     25.96     32.45     0.535     0.236   0.71750
     32.45     38.94     0.623     0.236   0.74958
     38.94     45.43     0.690     0.236   0.77854
     45.43     51.92     0.732     0.236   0.81046
     51.92     58.41     0.753     0.236   0.83186
     58.41     64.91     0.759     0.236   0.84185
     64.91     71.39     0.762     0.236   0.84259
     71.39     77.88     0.762     0.236   0.83290
     77.88     84.37     0.757     0.236   0.80238
     84.37     90.86     0.747     0.236   0.75198
     90.86     97.35     0.725     0.236   0.69326
     97.35    103.84     0.701     0.236   0.60511
    103.84    110.33     0.664     0.155   0.71014
    110.33    116.82     0.621     0.093   0.82180
    116.82    123.31     0.560     0.056   0.84329
    123.31    129.80     0.465     0.024   0.83398
EBMAS
   43
   1    2.017E3
   2    2.017E3
   3    2.017E3
   4    2.017E3
   5    2.017E3
   6    2.017E3
   7    2.017E3
   8    2.017E3
   9   76.374E3
  10   76.374E3
  11   81.148E3
  12   81.148E3
  13  154.005E3
  14  110.395E3
  15  136.827E3
  16  136.827E3
  17  269.548E3
  18  162.069E3
  19  162.069E3
  20  118.878E3
  21  118.878E3
  22  132.323E3
  23  132.323E3
```

PREFX.HVCR (cont.)

24	385.498E3
25	148.427E3
26	148.427E3
27	126.816E3
28	126.816E3
29	107.208E3
30	107.208E3
31	205.088E3
32	162.963E3
33	109.803E3
34	109.803E3
35	86.105E3
36	86.105E3
37	53.910E3
38	53.910E3
39	41.950E3
40	41.950E3
41	22.925E3
42	22.925E3
43	30.688E3

APPENDIX F

VASDAT SAMPLE RUNS

```
@RUN VASDAT

        V   V    A     SSS   DDDD     A    TTTTT
        V   V   A A   S   S  D   D   A A     T
        V   V  A   A  S      D   D  A   A    T
        V   V  AAAAA   SSS   D   D  AAAAA    T
        V   V  A   A      S  D   D  A   A    T
         V V   A   A  S   S  D   D  A   A    T
          V    A   A   SSS   DDDD   A   A    T


ENTER A 5 CHARACTER PREFIX NAME TO IDENTIFY DATA FILES
CDTST


A CONTROL FILE EXISTS WITH THE NAME CDTST.USE.
IS A NEW PREFIX TO BE USED TO IDENTIFY THE FILES FOR THIS ANALYSIS:
          1= YES
          0= NO
0

IS CONTROL OVER MASTER CONTROL CODES REQUIRED:
          1= YES - recommended for experienced users only
          0= NO
0

ENTER TYPE OF ANALYSIS TO BE PERFORMED:
          1= GEOMETRY, ELEMENT DATA CHECK AND MODEL INTEGRITY TEST
          2= LOAD DATA CHECK
          3= STATIC ANALYSIS
          4= BUCKLING ANALYSIS
          5= NATURAL FREQUENCY
          6= TIME HISTORY DYNAMIC RESPONSE
          7= RESPONSE SPECTRUM ANALYSIS
          8= FREQUENCY RESPONSE
          9= SUPPORT MOTION
         10= ERROR ESTIMATION
3

GEOMETRY GENERATION SECTION

GEOMETRY DATA EXISTS ON FILE CDTST.GOM.
CHOOSE ONE OF THE FOLLOWING OPTIONS:
          0= USE EXISTING DATA
          1= REGENERATE DATA
          2= MODIFY EXISTING DATA
0
```

BOUNDARY CONDITION/STIFFNESS MODIFICATION SECTION

```
BOUNDARY CONDITIONS/STIFFNESS MODIFICATIONS EXIST ON FILE CDTST.SMD.
CHOOSE ONE OF THE FOLLOWING OPTIONS:
          0= USE EXISTING DATA
          1= REGENERATE DATA
          2= MODIFY EXISTING DATA
0
```

LOAD GENERATION SECTION

```
LOAD DATA EXISTS ON FILE CDTST.LOD.
CHOOSE ONE OF THE FOLLOWING OPTIONS:
          0= USE EXISTING DATA
          1= REGENERATE DATA
          2= MODIFY EXISTING DATA
0
```

DISPLACEMENT SECTION

- control over printing output only

ELEMENT STRESSES SECTION

- control over printing output only

```
CONTROL DATA HAS BEEN STORED ON FILE CDTST.USE
GEOMETRY DATA HAS BEEN STORED ON FILE CDTST.GOM
STIFFNESS MODIFICATION DATA HAS BEEN STORED ON FILE CDTST.SMD
LOAD DATA HAS BEEN STORED ON FILE CDTST.LOD
```

```
@RUN VASDAT

     V   V    A     SSS   DDDD     A    TTTTT
     V   V   A A   S   S  D   D   A A     T
     V   V  A   A  S      D   D  A   A    T
     V   V  AAAAA   SSS   D   D  AAAAA    T
     V   V  A   A      S  D   D  A   A    T
      V V   A   A  S   S  D   D  A   A    T
       V    A   A   SSS   DDDD   A   A    T
```

ENTER A 5 CHARACTER PREFIX NAME TO IDENTIFY DATA FILES
CDTST

A USE FILE EXISTS WITH THE NAME CDTST.USE.
IS A NEW PREFIX TO BE USED TO IDENTIFY THE FILES FOR THIS ANALYSIS:
1= YES
0= NO
0

IS CONTROL OVER MASTER CONTROL CODES REQUIRED:
1= YES - RECOMMENDED FOR EXPERIENCED USERS ONLY
0= NO
0

IS A SUMMARY OF EXISTING DATA FILES REQUIRED:
1= YES
0= NO
1

SUMMARY:
GEOMETRY FILE EXISTS (PREFX.GOM)
STIFFNESS MATRIX MODIFICATION FILE EXISTS (PREFX.SMD)
USE FILE EXISTS (PREFX.USE)
LOAD FILE EXISTS (PREFX.LOD)

MASTER CONTROL CODES

MASTER CONTROL CODES WILL BE USED TO DETERMINE THE ANALYSIS OPTIONS WHICH WILL BE ACTIVATED IN VAST. IN RESPONSE TO EACH OF THE FOLLOWING PROMPTS, INPUT THE APPROPRIATE VALUE.

GEOMETRY AND ELEMENT GENERATION CODE

INPUT IELEMS:

0= RESTART (ELEMENT MATRICES HAVE BEEN COMPUTED DURING A PREVIOUS RUN.

1= ELEMENT MATRICES ARE COMPUTED (BOTH REGULAR ELEMENTS AND SUPERELEMENTS IF SUBSTRUCTURING IS USED).
2= ONLY REGULAR ELEMENT MATRICES ARE COMPUTED. THIS APPLIES ONLY WHEN SUBSTRUCTURING IS USED, AND PERMITS THE USER TO COMPUTE REGULAR ELEMENTS AND SUPERLEMENTS IN TWO STEPS. THE FEATURE PERMITS GEOMETRY AND REGULAR ELEMENT DATA TO BE CHECKED PRIOR TO THE MORE COSTLY COMPUTATION OF SUPERELEMENT MATRICES.
3= RESTART FOR COMPUTING SUPERELEMENT MATRICES. THIS APPLIES ONLY WHEN SUBSTRUCTURING IS USED, AND ASSUMES THAT A PREVIOUS RUN WITH IELEMS = 2 HAS BEEN MADE.
4= GROUPS OF SUBSTRUCTURE AND SUPERELEMENT FILES HAVE BEEN GENERATED BY PREVIOUS EXECUTIONS AND ARE TO BE COMBINED INTO ONE GROUP OF FILES.

0

<u>BANDWIDTH REDUCTION CODE</u>

INPUT IBANRD:

≠0 NODES ARE RENUMBERED TO REDUCE BANDWIDTH (FOR SUBSTRUCTURE ANALYSIS, THE MASTER NODES ONLY ARE RENUMBERED).

0

<u>MATRIX ASSEMBLY CODE</u>

INPUT IASSEM:

≠0 ELEMENT STIFFNESS AND MASS MATRICES ARE ASSEMBLED (FOR SUBSTRUCTURE ANALYSIS, THIS APPLIES TO SUPERELEMENT MATRICES).

1

<u>STIFFNESS MODIFICATIONS CODE</u>

INPUT ISTIFM:

≠0 STIFFNESS MODIFICATION STEP IS ACTIVATED TO INCLUDE BOUNDARY CONDITIONS AND/OR STIFFNESS ADDITIONS AND/OR MULTI-POINT CONSTRAINTS. <u>NOTE: THIS STEP MUST BE PERFORMED FOR ALL TYPES OF ANALYSIS. IF THERE ARE NO STIFFNESS MODIFICATIONS TO BE MADE, IT MUST BE INDICATED IN THE INPUT DATA OF SECTION C6, NOT BE SETTING ISTIFM=0</u>.

1

<u>MASS MODIFICATIONS CODE</u>

INPUT IMASSM:

≠0 MASS MODIFICATION STEP IS ACTIVATED TO INCLUDE LUMPED MASSES AND/OR FLUID ADDED MASS. <u>NOTE: THIS STEP MUST BE PERFORMED FOR ALL TYPES OF ANALYSIS REQUIRING MASS (SEE NOTE 2, PAGE C3-1). IF THERE ARE NO MASS MODIFICATIONS TO BE MADE, IT MUST BE INDICATED IN THE INPUT DATA OF SECTION C7, NOT BE SETTING IMASSM=0</u>.

1= THE FLUID ADDED MASS (IF TO BE INCLUDED) IS ASSUMED TO BE AVAILABLE ON DISK FILE PREFX.T36.
2= THE FLUID ADDED MASS IS GENERATED BY VAST. DATA INPUT DESCRIBED IN SECTION D MUST BE PROVIDED BY THE USER. THE FLUID ADDED MASS IS STORED ON DISK FILE PREFX.T36.
3= SAME AS IMASSM=2, EXCEPT THAT EXECUTION OF VAST IS TERMINATED AS SOON AS DISK FILE PREFX.T36 HAS BEEN CREATED.
4= SAME AS IMASS=3 EXCEPT THAT THE FOUR FLUID ADDED MASS
(5) MODULES ARE EXECUTED ONE AT A TIME, I.E., IMASSM=4,5,6, AND
(6) 7 CAUSES, RESPECTIVELY, THE MODULES ADMAS1, ADMAS2, ADMAS3,
(7) AND ADMAS4 TO BE EXECUTED.

0

MATRIX DECOMPOSITION CODE

INPUT IDECOM:

≠0 MATRIX DECOMPOSITION IS PERFORMED. A MATRIX DECOMPOSITION IS REQUIRED FOR ALL TYPES OF ANALYSES. HOWEVER, THE FORM OF THE MATRIX TO BE DECOMPOSED DEPENDS ON THE TYPE OF ANALYSIS.

1

EIGENVALUE ANALYSIS CODE

INPUT IEIGEN:

≠0 EIGENVALUE ANALYSIS IS PERFORMED (SEE NOTES IN SECTION C9).
1= NATURAL FREQUENCIES OR BUCKLING LOAD DETERMINED VIA DIRECT ITERATION METHOD.
2= NATURAL FREQUENCIES OR BUCKLING LOAD DETERMINED VIA SUBSPACE ITERATION METHOD.
3= NUMERICAL ERROR ESTIMATION

0

LOAD CODE

INPUT ILOADS:

≠0 LOAD VECTORS FOR STATIC, DYNAMIC, OR FREQUENCY RESPONSE ANALYSIS ARE COMPUTED.
1= ELEMENT/CONCENTRATED.
2= SUPPORT MOTION (TRANSLATIONAL ACCELERATIONS).

1

DISPLACEMENT CODE

INPUT IDISPS:

≠0 NODAL POINT DISPLACEMENTS ARE COMPUTED (SEE NOTES IN SECTION C11).
1= STATIC DISPLACEMENTS.
2= DYNAMIC DISPLACEMENTS VIA MODAL SUPERPOSITION.
3= DYNAMIC DISPLACEMENTS VIA DIRECT INTEGRATION.
4= RESPONSE SPECTRUM.
5= FREQUENCY RESPONSE.

1

ELEMENT STRESS CODE

```
INPUT ISTRES:
          ≠0 ELEMENT STRESSES ARE COMPUTED.
1
```

MASTER CONTROL CODE DEFINITION IS COMPLETE

MATRIX ASSEMBLY SECTION

- exact response to be determined later

BOUNDARY CONDITION/STIFFNESS MODIFICATION SECTION

```
BOUNDARY CONDITIONS/STIFFNESS MODIFICATIONS EXIST ON FILE CDTST.SMD.
CHOOSE ONE OF THE FOLLOWING OPTIONS:
          0= USE EXISTING DATA
          1= REGENERATE DATA
          2= MODIFY EXISTING DATA
0
```

MATRIX DECOMPOSITION SECTION

- exact response to be determined later

LOAD GENERATION SECTION

```
LOAD DATA EXISTS ON FILE CDTST.LOD.
CHOOSE ONE OF THE FOLLOWING OPTIONS:
          0= USE EXISTING DATA
          1= REGNERATE DATA
          2= MODIFY EXISTING DATA
0
```

DISPLACEMENT SECTION

- exact response to be determined later

ELEMENT STRESSESS SECTION

- exact response to be determined later

```
CONTROL DATA HAS BEEN STORED ON FILE CDTST.USE.
GEOMETRY DATA HAS BEEN SOTRED ON FILE CDTST.GOM.
STIFFNESS MODIFICATION DATA HAS BEEN STORED ON FILE CDTST.SMD.
LOAD DATA HAS BEEN STORED ON FILE CDTST.LOD.
```

APPENDIX G

PROGRAM LOADGT

LOADGT

LOADGT provides the means to generate a VAST load file for dynamic concentrated load time histories using a graphics digitizer. The load history plot is placed on the digitizer tablet, appropriately scaled and then converted to a VAST load file using the tablet pen. Load magnitude and time is computed automatically. No calculations by the user are required. Separate or superimposed load history plots may be used. Each loading function is digitized according to node and degree-of-freedom, which are inputted via the terminal keyboard. Computer plots of the digitized data can be obtained using the program GPLOT, which can be called directly within LOADGT.

The program requires no special files as input. The program generates a VAST PREFX.LOD file.

Operating Instructions for LOADGT

Hardware Required: TEKTRONIX 4953 GRAPHICS TABLET
TEKTRONIX 4954 GRAPHICS TABLET
TEKTRONIX 4956 GRAPHICS TABLET

Software Required: FORTRAN COMPILER
PLOT10 GRAPHICS PACKAGE
GPLOT
PLOTV

Command Sequence:

(i) Compile source program (proceed to (ii) if LOADGT.OBJ exists)

```
@ FORTRA <CR>
* FORTRA <LOADGT.MTL/BINARY : LOADGT.OBJ <CR>
@ <CONTROL C>
```

(ii) Load Complied Program

```
@ LOAD DREA : <23F> LOADGT.OBJ/REL, DREA : <23F>
  GPLOT.054/REL, DREA : <23F> PLOTV.054/REL, DS :
  <LIB> PLOT10/LIB <CR>
```

(iii) Executing Program

```
@ START <CR>
```

TERMINAL RESPONSE:

(1) SUBROUTINE LOADGT PROGRAM TO AUTOMATICALLY GENERATE
A DYNAMIC LOAD FILE FOR VAST ANALYSIS USING A GRAPHICS TABLET.

(2) ENTER VAST PREFIX FILE NAME.

(3) WHAT IS THE LINE SPEED?

(4) IDENTIFY TERMINAL TYPE ACCORDING TO RESOLUTION,
CURSOR AND COLOUR CAPACITY:

ENTER 0 FOR TEKTRONIX 4006 OR EQUAL (LOW RES/NO CURS/NO COL)
1 FOR TEKTRONIX 401* OR EQUAL (LOW RES/CURSORS/NO COL)
2 FOR TEKTRONIX 4014/15 (HI RES/CURSORS/NO COL)
3 FOR TEKTRONIX 4113 (COLOR):

(5) IDENTIFY SPECIAL HARDWARE REQUIREMENTS:

ENTER 0 IF NO EXTRA EQUIPMENT IS TO BE USED
1 FOR TEKTRONIX 4907 FILE MANAGER
2 FOR TEKTRONIX 4662 PLOTTER
3 FOR TEKTRONIX 4663 PLOTTER
4 FOR CALCOMP 1012 PLOTTER

(6) DOES THIS TERMINAL HAVE A GRAPHICS TABLET?

(0 = NO 3 = 4953 TABLET 4 = 4954 OR 4956 TABLET

(7) ENTER 1 TO TEST THE TABLET
0 TO CONTINUE

When using the graphics tablet initially the user should test the tablet. If the user inputs a 0, response (8) appears.

(7a) BEGIN SELECTING POINTS
AFTER EACH COORDINATE IS PRINTED

ENTER 0 TO CONTINUE OR 1 TO RETURN

A cursor will appear on the screen. Using the tablet pen the user can check the tablet for accuracy.

(8) ENTER THE NUMBER OF NODES TO BE LOADED

Separate or super-imposed plots of loading functions may be used with this program. Each loading function is digitized according to node and corresponding degree of freedom which is inputted via the terminal keyboard.

(9a) INPUT THE NODE NUMBER FOR THIS LOADING FUNCTION

(9b) INPUT THE DEGREE OF FREEDOM FOR THIS LOADING FUNCTION

(9c) *** TO END LOADING FOR THIS NODE ENTER A ZERO ***

This prompt only appears after the loading function for the first D.O.F. of any node has been digitized.

(10a) ** TABLET SETTINGS **

SPECIFY THE FOLLOWING POINTS ON THE GRAPHICS TABLET

1 - LOWER LEFT CORNER OF THE *COMMAND* BOX
2 - UPPER RIGHT CORNER OF THE *COMMAND* BOX

The command box should be designed as in Figure 1 and placed at any location on the graphics tablet. Dimension L can be any size.

(10b) 3 - LOWER LEFT CORNER OF DRAWING
4 - UPPER RIGHT CORNER OF DRAWING
5 - LOCAL ORIGIN
6 - A POINT ALONG THE X AXIS TO DEFINE THE SCALE
7 - A POINT ALONG THE Y AXIS TO DEFINE THE SCALE

If separate plots are used to define the loading functions (Figure 2) of two degrees of freedom for one node, points 3-7 will have to be defined for each plot. If the plots are super-imposed as in Figure 3, points 3-7 can be defined using the same points for each loading function.

(11) ENTER THE X COORDINATE FROM THE LOCAL ORIGIN TO THE SCALING POINT (POINT #6)

(12) ENTER THE Y COORDINATE FROM THE LOCAL ORIGIN TO THE SCALING POINT (POINT #7)

(13) BEGIN ENTERING DATA POINTS ON THE LOADING FUNCTION TO STOP PRESS "STOP" IN THE COMMAND BOX.

A cursor will appear on the screen. Using the tablet pen, the user will input NTIME data points on the loading function. The times used for each loading function must be the same.

(14) DO YOU WANT TO PLOT THE LOADING FUNCTION FOR THIS NODE AND D.O.F.

1 = YES
0 = NO

An input of 0 transfers the terminal to response #(96). Prompts 15 to 35 appear only if plotting is desired.

(15) PROMPTS (3), (4) and (5) reappear

(16) PROGRAM GPLOT: GENERAL PURPOSE PROGRAM FOR
PLOTTING A SET OF DATA POINTS

(17) ARE ALL DATA SETS TO BE PLOTTED? (0/1 = N/Y)

(18) DO YOU WANT A LIST OF THE SPECIFIED DATA SETS? (0/1 = N/Y)

(19) DO YOU WANT TO MAKE ANY MODIFICATIONS (0/1 = N/Y)

(20) ENTERING 0 IN RESPONSE TO ALL REMAINING PROMPTS WILL MINIMIZE INPUT REQUIRED AND GENERATE DEFAULT PLOT

(21) DO YOU WANT TO SUPPRESS THE LEGEND ON THE PLOTS? (0/1 = N/Y)

In a default plot, GPLOT reserves a space at the right hand side of the plot for a legend. The legend contains the lable which identifies the data set from Record #6 (see Figure 1) and a symbol code to identify the associated curve which is plotted. This prompt allows the user to suppress this legend.

(22) SPECIFY PLOT LINE TYPE (0 = SOLID; 1 = DASHED):

The lines joining the data points may be either solid or dashed, according to how the user responds to this prompt.

(23) ENTER 0 TO PLOT THE X DATA ON THE HORIZONTAL AXIS
OR 1 TO PLOT THE X DATA ON THE VERTICAL AXIS:

This prompt controls whether the X data set is to be associated with the horizontal or vertical axis on the plot, and consequently, whether the Y data set is to be associated with the vertical or horizontal axis.

(24) ENTER 0 TO PLOT THE X DATA ON A LINEAR SCALE
OR 1 TO PLOT THE X DATA ON A LOG SCALE:

(25) ENTER 0 TO PLOT THE Y DATA ON A LINEAR SCALE
OR 1 TO PLOT THE Y DATA ON A LOG SCALE:

These prompts appear only if all of the points of the associated data sets which are to be plotted contain values greater than 0. If this is the case, the user may plot that data set on either a linear or log scale.

(26) DO YOU WANT TO MODIFY THE X DATA WINDOW? (0/1 = N/Y)

(27) DO YOU WANT TO MODIFY THE Y DATA WINDOW? (0/1 = N/Y)

GPLOT automatically determines, from the data sets to be plotted, the minimum and maximum values of the X and Y data points. These minimum and maximum values define a data 'window' which is used by PLOT10 in scaling the plot. These prompts allow the user to define either X or Y data window, which may be either smaller or larger than that determined automatically. If the user chooses to modify the data window, the calculated minimum and maximum limits of the window are presented and the user is prompted for values to be used instead of these calculated values.

(28) DO YOU WANT TO MODIFY THE HORIZONTAL SCREEN WINDOW? (0/1 = N/Y)

(29) DO YOU WANT TO MODIFY THE VERTICAL SCREEN WINDOW? (0/1 = N/Y)

Similar to the data 'window', PLOT10 also works with a 'screen window', which defines the limits on the screen of the plot. GPLOT incorporates a default screen window, of which either dimension may be altered, through these prompts. If the user chooses to modify

the screen window, the present limits of the window appear, and the user is prompted for new values. Note that the user may only define the window to be smaller than the default, not larger.

(30) DO YOU WANT TO MOVE THE HORIZONTAL DATA LABEL? (0/1 = N/Y)

(31) ENTER VERTICAL OFFSET FACTOR FOR HORIZONTAL DATA LABEL:

(32) DO YOU WANT TO MOVE THE VERTICAL DATA LABEL? (0/1 = N/Y)

(33) ENTER HORIZONTAL OFFSET FACTOR FOR VERTICAL DATA LABEL:

These prompts allow the user control over the position of the axis labels. The user may specify, in the form of an offset factor, how far the label is from either the horizontal or vertical axis. An offset factor of 1 is used by default in GPLOT. An offset factor of 0 will place the axis label right on the axis and an offset factor of -1 will move the label into the plotting area.

(34) AFTER PLOTTING USE:

W - WINDOW
R - TO RECOVER ORIGINAL
PRESS "RETURN" TO PLOT.

The interactive user input has concluded and when the user enters 'RETURN', the plot will appear on the screen. Once the plot appears, the user will have several options. If the terminal has cursors, the user may 'window' the plot using the cursor to define the lower left and upper right hand corners of the plot. The intention to window using the cursor is indicated by entering 'W' with a carriage return after the plot appears. Otherwise, the user simply enters 'RETURN' and the GPLOT will allow either termination

or redefinition of the plot, starting with definition of the data sets to be plotted.

(35) ENTER 0 TO TERMINATE
1 TO CONTINUE.

After plotting the loading function for the specified node and D.O.F., prompt (9b) appears. If no other load function for this node is to be digitized enter a zero. If a zero is entered in response to prompt (9b), the terminal transfers to:

(i) response (9a); for digitization of load curves for the second and subsequent nodes to be loaded
(ii) response (36); after the digitization of the load curve for the last node to be loaded.

(36) OUTPUT FILE PREFX.LOD HAS BEEN CREATED.

APPENDIX H

BIBLIOGRAPHY ON THE BOUNDARY ELEMENT METHOD FOR PLATE BENDING

BIBLIOGRAPHY ON THE BOUNDARY ELEMENT METHOD FOR PLATE BENDING

1. N.J. ALTIERO and D.L. SIKARSKIE. A boundary integral method applied to plates of arbitrary plan form, Computers and Structures, Vol. 9, pp. 163-168 (1978).

2. H. ANTES. On a regular boundary integral equation and a modified Trefftz method in Reissner's plate theory, Eng. Anal., Vol. 1, No. 3, pp. 149-153 (September 1984) [Reissner's plate theory].

3. L.D. BALANI and S. RAWTANI. Free vibrations of a circular plate clamped on part of its boundary and simply supported on the remainder. J. Inst. Eng. India, Part ME, Vol. 66, pp. 25-29 (July 1985).

4. J. BALAS, V. SLADEK and J. SLADEK. Boundary integral equation method for plates resting on a two-parameter foundation. Z. Angew. Math. Mech., Vol. 64, No. 3, pp. 137-146 (1984) [elastic foundation; soil-structure interaction; thin plates].

5. A.M. BERGER. A new approach to the analysis of large deflections of plates. J. Appl. Mech., Vol. 22, pp. 465-??? (1955).

6. G. BEZINE. Boundary integral formulation for plate flexure with arbitrary boundary conditions. Mech. Res. Comm., Vol. 5, No. 4, pp. 197-206 (1978).

7. G.P. BEZINE and D.A. GAMBY. A new integral equation formulation for plate bending problems, pp. 327-342. In "Recent Advances in Boundary Element Methods, Ed. by C.A. BREBBIA, Pentech Press, London, Plymouth (1978).

8. G. BEZINE. A mixed boundary-integral-finite element approach to plate vibration problem. Mech. Res. Comm., Vol. 7, No. 3, pp. 141-150 (1980).

9. G. BEZINE. Application of similarity to research of new boundary integral equations for plate flexure problems. Appl. Math. Modelling, Vol. 5, No. 2, pp. 66-70 (April 1981).

10. G. BEZINE. Boundary integral equation method for plate flexure with conditions inside the domain. Int. J. Numer. Methods Engng., Vol. 17, No. 11, pp. 1647-1657 (November 1981).

11. G. BEZINE and D. FORTUNE. Contact between plates by a new direct boundary integral equation formulation. Int. J. Solids Structures, Vol. 20, No. 8, pp. 739-746 (1984) [plate theory].

12. G. BEZINE and D. BONNEAU. On a procedure for numerical evaluation of surface integrals in two-dimensional boundary integral equation method. Eng. Anal., Vol. 2, No. 1, pp. 2-8 (March 1985) [plate flexure; boundary integral formulation].

13. G. BEZINE, A. CIMETIERE and J.P. GELBERT. Unilateral buckling of thin elastic plates by the boundary integral equation method. Int. J. Numer. Methods in Engng., Vol. 21, No. 12, pp. 2189-2199 (December 1985) [boundary integral equations; thin plates].

14. W. CECOT and J. ORKISZ. Approximation of boundary values in the boundary element method. Appl. Math. Modelling, Vol. 8, No. 1, pp. 23-26 (February 1984) [plates].

15. Y.Z. CHEN. Special boundary-element formulation for multiple-circular-hole problems in an infinite plate. Computer Methods Appl. Mech. Eng., Vol. 50, No. 3, pp. 263-273 (August 1985).

16. J.A. COSTA and C.A. BREBBIA. Plate bending using BEM, pp. 43-63, Vol. 3, In "Boundary Elements VI", CML Publications/Springer-Verlag (1984).

17. J.A. COSTA and C.A. BREBBIA. Boundary element method applied to plates on elastic foundations, Engineering Analysis, Vol. 2, No. 4, pp. 174-183 (December 1985).

18. T.A. CRUISE and E.Z. POLCH. Elastoplastic BIE analysis of cracked plates and related problems. Part 1: Formulation. Int. J. Numer. Methods in Engng., Vol. 23, No. 3, pp. 429-437 (March 1986) [plates; elastoplasticity; fracture mechanics; boundary integral equations].

19. T.A. CRUISE and E.Z. POLCH. Elastoplastic BIE analysis of cracked plates and related problems. Part 2: Numerical Results. Int. J. Numer. Methods Engng., Vol. 23, No. 3, pp. 439-452 (March 1986) [plates; elastoplasticity; fracture mechanics; boundary integral equations].

20. J. DIAZ DEL VALLE and A. SUMARTIN. Comparative analysis between FEM and boundary methods in bending of plates, Eng. Anal., Vol. 1, No. 2, pp. 88-91 (1984) [plate bending; mathematical techniques; Poisson-Kirchoff assumption].

21. Q.H. DU and Z.H. YAO. Applications of the boundary element method to two and three dimensional stress analysis and plate bending problems in elasticity, pp. 269-281, In "Boundary Elements in Engineering", Proceedings of the Fourth International Seminar on Boundary Element Methods held at Southampton University, England in September 1982, Ed. by C.A. BREBBIA, Springer-Verlag, Berlin and New York (1982).

22. Q. DU, Z. YAO and G. SONG. Solution of some plate bending problems using the boundary element method, Appl. Math. Modelling, Vol. 8, No. 1, pp. 15-22 (1984) [Kirchoff theory of thin plate bending].

23. H.J. FLETCHER and C.J. THORNE. Thin rectangular plates on elastic foundation. ASME Journal of Applied Mechanics, Vol. 19, pp. 361-368 (1952).

24. D.J. FORBES and A.R. ROBINSON. Numerical analysis of elastic plates and shallow shells by an integral equation method. University of Illinois, Structural Research Series Report 345 (1969).

25. T. FUKUI. On corner solutions by indirect B.I.E.M., pp. 929-938, Boundary Elements, Proceedings of the 5th International Conference held in Hiroshima, Japan in November 1983. Published by Springer-Verlag, West Germany and New York, USA (1983).

26. G. GOSPODINOV and D. LJUTSKANOV. The boundary element method applied to plates, Appl. Math. Modelling, Vol. 6, No. 4, pp. 237-244 (1982) [thin elastic plates; dynamic analysis; stability analysis].

27. S. GUO-SHU and S. MUKHERJEE. Boundary element method analysis of bending of elastic plates of arbitrary shape with general boundary conditions, Eng. Anal., Vol. 3, No. 1, pp. 36-44 (March 1986).

28. E.B. HANSEN. Numerical solutions of integro-differential and singular integral equations for plate bending problems. J. Elasticity, Vol. 6, No. 1, pp. 39-56 (1976).

29. H. IRSCHIK. Boundary-integral equation method for bending of orthotropic plates. Int. J. Solids Struct., Vol. 20, No. 3, pp. 245-255 (1984) [plate bending; orthotropic linear plates].

30. M.A. JASWON, M. MAITI and G.T. SYMM. Numerical harmonic analysis and some applications. Int. J. Solids Structures, Vol. 3, pp. 309-332 (1967).

31. M.A. JASWON and M. MAITI. An integral equation formulation of plate bending problems. Journal of Engineering Mathematics, Vol. 2, No. 1, pp. 83-93 (January 1968).

32. N. KAMIYA and Y. SAWAKI. An integral equation approach to finite deflection of elastic plates, Int. J. Non-Linear Mechanics, Vol. 17, No. 3, pp. 187-194 (1982).

33. N. KAMIYA and Y. SAWAKI. General boundary element method for bending analysis of orthotropic elastic plates. Res. Mech., Vol. 5, No. 4, pp. 329-334 (December 1982).

34. N. KAMIYA and Y. SAWAKI. Simplified nonlinear boundary analysis of flat plates and shallow shells by boundary element approach based on Berger equation, pp. 289-297. Proceedings of Conference on "Numerical Methods in Industrial Forming Processes" held in Swansea, Wales on 12-16 July 1982. Published by Pineridge Press, Swansea, Wales (1982).

35. N. KAMIYA, Y. SAWAKI and Y. NAKAMURA. Boundary element nonlinear bending analyses of clamped sandwich plates and shells, pp. 515-523, In "Boundary Elements in Engineering", Proceedings of the Fourth International Seminar on Boundary Element Methods held at

Southampton University, England in September 1982, Ed. by C.A. BREBBIA, Springer-Verlag, Berlin and New York (1982) [plate bending].

36. N. KAMIYA, Y. SAWAKI, Y. NAKAMURA and A. FUKUI. An approximate finite deflection analysis of a heated elastic plate by the boundary element method, Appl. Math. Modelling, Vol. 6, No. 1, pp. 23-27 (1982) [thermoelasticity].

37. N. KAMIYA, Y. SAWAKI and Y. NAKAMURA. Post-buckling analysis by the boundary element method. Eng. Anal., Vol. 1, No. 1, pp. 40-44 (March 1983) [post-buckling deformation; heated plates; boundary element methods].

38. N. KAMIYA. Boundary element bending analysis of heated elastic plates. Appl. Math. Modelling, Vol. 7, No. 5, pp. 377-379 (October 1983) [plate bending; thermal effects].

39. N. KAMIYA and Y. SAWAKI. Boundary integral formulation for some inhomogeneous differential equations. Eng. Anal., Vol. 1, No. 4, pp. 188-194 (December 1984) [plate bending; heat transfer].

40. N. KAMIYA and Y. SAWAKI. Simplified large deflection analysis of plates resting on elastic foundation. Rev. Roum. Sci. Tech. Ser. Mech. Appl., Vol. 3, No. 1, pp. 91-99 (January-February 1985).

41. T. KATAYAMA, T. SEKIYA and H. TAI. Bending analysis of perforated plates with unsupported or clamped edges by BEM. pp. 517-526. Boundary Elements V, Proceedings of the 5th International Conference held in Hiroshima, Japan in November 1983. Published by Springer-Verlag, Berlin and New York (1983).

42. J.T. KATSIKADELIS. The analysis of plates on elastic foundation by the boundary integral equation method, Ph.D. Dissertation, Polytechnic Institute of New York (1982).

43. J.T. KATSIKADELIS and A.E. ARMENAKAS. Plates on elastic foundation by BIE method. J. Eng. Mech., Vol. 110, No. 7, pp. 1086-1105 (July 1984) [boundary integral equations; elastic foundations].

44. J.T. KATSIKADELIS and A.E. ARMENAKAS. Analysis of clamped plates on elastic foundation by the boundary integral equation method. Paper 84-WA/APM-6, Presented at the Winter Annual Meeting of the American Society of Mechanical Engineers held in New Orleans, La., USA 9-14, December 1984. Published by ASME, New York, 7 p (1984).

45. F.S. KELLEY. Mesh requirements for the analysis of a stress concentration by the specific boundary displacement method, pp. 39-42, Vol. 3. Computers in Engineering 1982: Proceedings of the 2nd International Computer Engineering Conference held in San Diego, California, USA on 15-19 August 1982. Published by ASME, New York, NY, USA (1982).

46. K. KISHIMOTO, I. YAMAGUCHI, M. TACHIHARA, S. AOKI and M. SAKATA. Elastic-plastic fracture mechanics analysis by combination of boundary and finite element methods, pp. 975-984. Boundary Elements V. Proceedings of the 5th International Conference held in Hiroshima, Japan in November 1983. Published by Springer-Verlag, Berlin and New York (1983).

47. H. KISU, R. YUUKI and H. KITAGAWA. On the methods to determine the stress intensity factors in boundary element method, pp. 53-56. Proceedings of the 27th Japan Congress on Materials Research held in Tokyo, Japan in September 1983. Published by Society of Materials Science, Kyoto, Japan (1984).

48. M. KITAHARA. Boundary integral equation methods in eigenvalue problems in elastodynamics and thin plates. Elsevier, New York, 282 p (1985).

49. B. KRAYTERMAN and G.M. SABNIS. Geometrical nonlinear bending of plates and shells with consideration of loading history, pp. 714-717, Vol. 2, Proceedings of the 4th Engineering Mechanics Division Speciality Conference: Recent Advances in Engineering Mechanics and Their Impact on Civil Engineering Practice. West Lafayette, Indiana, USA 23-25 May 1983. Published by ASCE, New York, NY, USA (1983).

50. I.A. LEVITSKII and YU I. NAGDRNYI. Reaction of a plate with a hole to a dynamic load. Mech. Solids, Vol. 19, No. 3, pp. 168-171 (1984) [plates; dynamic response; boundary integral equations].

51. M. MAITI and S.K. CHAKRABARTY. Integral equation solutions for simply supported polygonal plates. Int. J. Engng. Sci., Vol. 12, No. 10, pp. 793-806 (October 1974).

52. R.D. MILLER. Comparison of the boundary element method and the finite element method for the solution of a uniformly loaded circular plate, pp. 521-539. BETECH86: Proceedings of 2nd Boundary Element Technology Conference held at the Massachusetts Institute of Technology, USA in June 1986. Ed. by J.J. CONNOR and C.A. BREBBIA. Published by Computational Mechanics Publications, Southampton, UK (1986).

53. M. MORJARIA and S. MUKHERJEE. Inelastic analysis of transverse deflection of plates by the boundary element method. J. Appl. Mech., Trans. ASME, Vol. 47, No. 2, pp. 291-296 (June 1980) [plates].

54. M. MORJARIA and S. MUKHERJEE. Time-dependent inelastic deformation of plates with cracks by the boundary element method. Int. J. Solids Struct., Vol. 17, No. 1, pp. 127-143 (1981) [fracture mechanics; plates; plasticity; creep].

55. M. MOUSSOUROS. Comparisons of buckling load predictions for a ring stiffened cylinder, pp. 395-398. Vol. 1, Proceedings of the 4th Engineering Mechanics Divisions Speciality Conference: Recent Advances in Engineering Mechanics and Their Impact on Civil Engineering Practice held in West Lafayette, Indiana, USA on 23-25 May 1983. Published by ASCE, New York, NY, USA (1983).

56. S. MUKHERJEE and M. MORJARIA. Boundary element formulation for planar time-dependent inelastic deformation of plates with cutouts. Int. J. Solids Structures, Vol. 17, No. 1, pp. 115-126 (1981) [circular plates; fracture mechanics].

57. S. MUKHERJEE and M. MORJARIA. Boundary element analysis of time-dependent inelastic deformation of cracked plates. Int. J. Solids Struct., Vol. 17, No. 8, pp. 753-763 (1981) [plates; stability; fracture; boundary integral methods].

58. S. MUKHERJEE. Bending of viscoplastic plates. Chapter 8, pp. 122-149, Boundary Element Methods in Creep and Fracture. By S. MUKHERJEE. Published by Applied Science Publishers, London and New York (1982).

59. H. OKADA, M. FUJIMOTO and H. ABE. Elastoplastic instability analysis of thin plates, pp. 589-597. Boundary Elements V. Proceedings of the 5th International Conference held in Hiroshima, Japan in November 1983. Published by Springer-Verlag, Berlin and New York (1983).

60. F. PARIS and S. de LEON. Simply supported plates by the boundary integral equation method. Int. J. Numer. Methods Engng., Vol. 23, No. 2, pp. 173-191 (February 1986) [Kirchoff plate bending; integral methods].

61. S.N. PLYAT and A.P. VILLASOR, Jr. Modal analysis of perforated plates with triangular patterns of circular holes, pp. 326-332. Proceedings of the 1st International Modal Analysis Conference and Exhibition held in Orlando, Fl., USA on 8-10 November 1982. Published by Union College, Schenectady, NY, USA (1982).

62. J. PUTTONEN and P. VARPASUO. Boundary element analysis of a plate on elastic foundations, Int. J. Numer. Methods in Engng., Vol. 23, No. 2, pp. 287-303 (February 1986).

63. QIN RONG. Fundamentals and applications of spline boundary element method, pp. 949-961. Boundary Elements V. Proceedings of the 5th International Conference held in Hiroshima, Japan in November 1983. Published by Springer-Verlag, Berlin and New York (1983).

64. R. SHAW. Elastic plate vibration by boundary integral equations - 1. Infinite Plates. Res. Mech. Vol. 4, No. 1, pp. 83-88 (January 1982) [plate vibrations; integral equations].

65. M. STERN. A general boundary integral formulation for numerical solution of plate bending problems, Int. J. Solids Struct., Vol. 15, No. 10, pp. 769-782 (October 1979).

66. M. STERN. Plate bending problems, Chapter 17, pp. 315-325, Boundary Element Techniques in Computer-Aided Engineering, Proceedings of the NATO Advanced Study on Boundary Elements in CAD, Southampton, England on 19-30 September 1983. Ed. by C.A. BREBBIA, Martinus Nijhoff Publishers, Dordrecht/Boston/Lanchaster (1984).

67. M. STERN. Boundary integral equations for bending of thin plates, Chapter 6, pp. 156-181. Progress in Boundary Element Methods - II, Ed. by C.A. BREBBIA, Pentech Press, London and Plymouth (1983) [plate bending; thin plate reciprocal work identity].

68. M. STERN. Formulating nonsingular boundary integral equations in linear elasticity, pp. 213-223. Advanced Topics in Boundary Elements Analysis. Ed. by T.A. CRUISE, A.B. PIFKO and H. ARMEN. Presented at the Winter Annual Meeting of the American Society of Mechanical Engineers in Miami Beach, Fl., USA on 17-22 November 1985. AMD Vol. 72, Published by ASME, New York, NY, USA (1985).

69. M. STERN and T.-L. LIN. Thin elastic plates in bending. In "Developments in Boundary Element Methods", Vol. 4, Elsevier Applied Science Publishers Ltd., (In Press) [boundary element methods; fundamental solutions; domain corners; transverse loading].

70. H. TAI, T. KATAYAMA and S. MOZUUMEMACHI. Boundary element method for bending of plates with free edges. Mechanics Research Communications, Vol. 10, pp. 121-??? (1983).

71. M. TANAKA. Integral equation approach to small and large displacements of thin elastic plates, pp. 526-539, In "Boundary Elements in Engineering", Proceedings of a Fourth International Seminar on Boundary Element Methods held at Southampton University, England in September 1982, Ed. by C.A. BREBBIA, Springer-Verlag, Berlin and New York (1982) [plate bending].

72. H. TOTTENHAM. The boundary element method for plates and shells, pp. 173-205, In "Developments in Boundary Element Methods-1", Ed. by P.K. BANERJEE, Applied Science Publishers Ltd., London (1979).

73. F. VANDER WEEEN. Application of the boundary integral equation method to Reissner's plate model. Int. J. Numer. Methods Engng., Vol. 18, pp. 1-10 (1982).

74. G.K.K. WONG and J.R. HUTCHINSON. An improved boundary element method for plate vibrations, pp. 272-289, Boundary Element Methods, Proceedings of 3rd International Seminar held at Irvine, California in 1981, Ed. by C.A. BREBBIA, Springer-Verlag, Berlin (1981) [elastodynamics].

75. B.C. WU and N.J. ALTIERO. A boundary integral method applied to plates and arbitrary plan form and arbitrary boundary conditions, Computers and Structures, Vol. 10, pp. 703-707 (1979).

76. CHEN XIANGFU and QIN RONG. Spline function used in boundary element methods, pp. 539-548. Boundary Elements V. Proceedings of the 5th International Conference held in Hiroshima, Japan in November 1983. Published by Springer-Verlag, Berlin and New York (1983).

77. Y. YAGAWA, T. YAMABUCHI and S. KITAGAMI. Infinite boundary element and its application to a combined finite-boundary element technique for unbounded field problem, pp. 1017-1026. Boundary Elements V. Proceedings of the 5th International Conference held in Hiroshima, Japan in November 1983. Published by Springer-Verlag, Berlin and New York (1983).

78. T.Q. YE and Y. LIU. Finite deflection analysis of elastic plate by the boundary element method. Appl. Math. Modelling, Vol. 9, No. 3, pp. 183-188 (June 1985).

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)
Martec Limited
1888 Brunswick Street, Suite 400
Halifax, NS B3J 3J8

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable).
UNCLASSIFIED

3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).
Simplified Models for Ship Analysis Final Report

4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)
M.E. Norwood et al

5. DATE OF PUBLICATION (month and year of publication of document)
Dec 92

6a. NO OF PAGES (total containing information include Annexes, Appendices, etc).
70

6b. NO. OF REFS (total cited in document)
78

7. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered).

8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include the address).
Defence Research Establishment Atlantic
9 Grove St., Dartmouth, Nova Scotia B2Y 3Z7

9a. PROJECT OR GRANT NO. (If appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant).

9b. CONTRACT NO. (If appropriate, the applicable number under which the document was written).
W7707-6-7739

10a. ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document).
TR-93-01

10b. OTHER DOCUMENT NOS. (Any other numbers which may be assigned this document either by the originator or by the sponsor).

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(X) Unlimited distribution
( ) Distribution limited to defence departments and defence contractors; further distribution only as approved
( ) Distribution limited to defence departments and Canadian defence contractors; further distribution only as approved
( ) Distribution limited to government departments and agencies; further distribution only as approved
( ) Distribution limited to defence departments; further distribution only as approved
( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in 11) is possible, a wider announcement audience may be selected).

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

DCD03 2/06/87-M

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

This report describes the work carried out under contract to the Defence Research Establishment Atlantic (DREA). Contract Serial No. OAE.86-0042, DSS File No. W7707-6-7739/01-OAE. This report is supported by two other documents produced under this contract (References 1 and 2).

5 04 The purpose of this work was to produce, through research and development, a capability for simplified analysis of ship structures. Simplified techniques for generating input data for finite element analysis have been developed. The work performed is described in Sections 2 to 12. //

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus. e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title).

ship structures

finite element analysis

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

#144581